FANUC Series 30*i*-MODEL B
FANUC Series 31*i*-MODEL B
FANUC Series 32*i*-MODEL B

Common to Lathe System / Machining Center System

OPERATOR'S MANUAL

B-64484EN/03

- No part of this manual may be reproduced in any form.
- All specifications and designs are subject to change without notice.

The products in this manual are controlled based on Japan's "Foreign Exchange and Foreign Trade Law". The export of Series 30*i*-B, Series 31*i*-B5 from Japan is subject to an export license by the government of Japan. Other models in this manual may also be subject to export controls.
Further, re-export to another country may be subject to the license of the government of the country from where the product is re-exported. Furthermore, the product may also be controlled by re-export regulations of the United States government.
Should you wish to export or re-export these products, please contact FANUC for advice.

The products in this manual are manufactured under strict quality control. However, when some serious accidents or losses are predicted due to a failure of the product, make adequate consideration for safety.

In this manual we have tried as much as possible to describe all the various matters.
However, we cannot describe all the matters which must not be done, or which cannot be done, because there are so many possibilities.
Therefore, matters which are not especially described as possible in this manual should be regarded as "impossible".

SAFETY PRECAUTIONS

This section describes the safety precautions related to the use of CNC units.
It is essential that these precautions be observed by users to ensure the safe operation of machines equipped with a CNC unit (all descriptions in this section assume this configuration). Note that some precautions are related only to specific functions, and thus may not be applicable to certain CNC units.
Users must also observe the safety precautions related to the machine, as described in the relevant manual supplied by the machine tool builder. Before attempting to operate the machine or create a program to control the operation of the machine, the operator must become fully familiar with the contents of this manual and relevant manual supplied by the machine tool builder.

CONTENTS

DEFINITION OF WARNING, CAUTION, AND NOTE s-1
GENERAL WARNINGS AND CAUTIONS s-2
WARNINGS AND CAUTIONS RELATED TO PROGRAMMING s-3
WARNINGS AND CAUTIONS RELATED TO HANDLING s-5
WARNINGS RELATED TO DAILY MAINTENANCE s-7

DEFINITION OF WARNING, CAUTION, AND NOTE

This manual includes safety precautions for protecting the user and preventing damage to the machine. Precautions are classified into **Warning** and **Caution** according to their bearing on safety. Also, supplementary information is described as a **Note**. Read the **Warning**, **Caution**, and **Note** thoroughly before attempting to use the machine.

⚠ WARNING
Applied when there is a danger of the user being injured or when there is a danger of both the user being injured and the equipment being damaged if the approved procedure is not observed.

⚠ CAUTION
Applied when there is a danger of the equipment being damaged, if the approved procedure is not observed.

NOTE
The Note is used to indicate supplementary information other than Warning and Caution.

- Read this manual carefully, and store it in a safe place.

GENERAL WARNINGS AND CAUTIONS

⚠ WARNING

1 Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

2 Before operating the machine, thoroughly check the entered data.
Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

3 Ensure that the specified feedrate is appropriate for the intended operation.
Generally, for each machine, there is a maximum allowable feedrate.
The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate.
If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

4 When using a tool compensation function, thoroughly check the direction and amount of compensation.
Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

5 The parameters for the CNC and PMC are factory-set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change.
Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

6 Immediately after switching on the power, do not touch any of the keys on the MDI panel until the position display or alarm screen appears on the CNC unit.
Some of the keys on the MDI panel are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.

7 The OPERATOR'S MANUAL and programming manual supplied with a CNC unit provide an overall description of the machine's functions, including any optional functions. Note that the optional functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.

8 Some functions may have been implemented at the request of the machine-tool builder. When using such functions, refer to the manual supplied by the machine-tool builder for details of their use and any related cautions.

⚠ CAUTION
The liquid-crystal display is manufactured with very precise fabrication technology. Some pixels may not be turned on or may remain on. This phenomenon is a common attribute of LCDs and is not a defect.

NOTE
Programs, parameters, and macro variables are stored in non-volatile memory in the CNC unit. Usually, they are retained even if the power is turned off.
Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from non-volatile memory as part of error recovery.
To guard against the occurrence of the above, and assure quick restoration of deleted data, backup all vital data, and keep the backup copy in a safe place.
The number of times to write machining programs to the non-volatile memory is limited.
You must use "High-speed program management" when registration and the deletion of the machining programs are frequently repeated in such case that the machining programs are automatically downloaded from a personal computer at each machining.
In "High-speed program management", the program is not saved to the non-volatile memory at registration, modification, or deletion of programs.

WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

This section covers the major safety precautions related to programming. Before attempting to perform programming, read the supplied OPERATOR'S MANUAL carefully such that you are fully familiar with their contents.

⚠ WARNING
1 **Coordinate system setting**
If a coordinate system is established incorrectly, the machine may behave unexpectedly as a result of the program issuing an otherwise valid move command. Such an unexpected operation may damage the tool, the machine itself, the workpiece, or cause injury to the user.
2 **Positioning by nonlinear interpolation**
When performing positioning by nonlinear interpolation (positioning by nonlinear movement between the start and end points), the tool path must be carefully confirmed before performing programming. Positioning involves rapid traverse. If the tool collides with the workpiece, it may damage the tool, the machine itself, the workpiece, or cause injury to the user.
3 **Function involving a rotation axis**
When programming polar coordinate interpolation or normal-direction (perpendicular) control, pay careful attention to the speed of the rotation axis. Incorrect programming may result in the rotation axis speed becoming excessively high, such that centrifugal force causes the chuck to lose its grip on the workpiece if the latter is not mounted securely. Such mishap is likely to damage the tool, the machine itself, the workpiece, or cause injury to the user.

⚠ WARNING

4 **Inch/metric conversion**
Switching between inch and metric inputs does not convert the measurement units of data such as the workpiece origin offset, parameter, and current position. Before starting the machine, therefore, determine which measurement units are being used. Attempting to perform an operation with invalid data specified may damage the tool, the machine itself, the workpiece, or cause injury to the user.

5 **Constant surface speed control**
When an axis subject to constant surface speed control approaches the origin of the workpiece coordinate system, the spindle speed may become excessively high. Therefore, it is necessary to specify a maximum allowable speed. Specifying the maximum allowable speed incorrectly may damage the tool, the machine itself, the workpiece, or cause injury to the user.

6 **Stroke check**
After switching on the power, perform a manual reference position return as required. Stroke check is not possible before manual reference position return is performed. Note that when stroke check is disabled, an alarm is not issued even if a stroke limit is exceeded, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

7 **Tool post interference check**
A tool post interference check is performed based on the tool data specified during automatic operation. If the tool specification does not match the tool actually being used, the interference check cannot be made correctly, possibly damaging the tool or the machine itself, or causing injury to the user. After switching on the power, or after selecting a tool post manually, always start automatic operation and specify the tool number of the tool to be used.

8 **Absolute/incremental mode**
If a program created with absolute values is run in incremental mode, or vice versa, the machine may behave unexpectedly.

9 **Plane selection**
If an incorrect plane is specified for circular interpolation, helical interpolation, or a canned cycle, the machine may behave unexpectedly. Refer to the descriptions of the respective functions for details.

10 **Torque limit skip**
Before attempting a torque limit skip, apply the torque limit. If a torque limit skip is specified without the torque limit actually being applied, a move command will be executed without performing a skip.

11 **Programmable mirror image**
Note that programmed operations vary considerably when a programmable mirror image is enabled.

12 **Compensation function**
If a command based on the machine coordinate system or a reference position return command is issued in compensation function mode, compensation is temporarily canceled, resulting in the unexpected behavior of the machine. Before issuing any of the above commands, therefore, always cancel compensation function mode.

WARNINGS AND CAUTIONS RELATED TO HANDLING

This section presents safety precautions related to the handling of machine tools. Before attempting to operate your machine, read the supplied OPERATOR'S MANUAL carefully, such that you are fully familiar with their contents.

⚠ WARNING

1 **Manual operation**
When operating the machine manually, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and feedrate have been specified correctly. Incorrect operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

2 **Manual reference position return**
After switching on the power, perform manual reference position return as required.
If the machine is operated without first performing manual reference position return, it may behave unexpectedly. Stroke check is not possible before manual reference position return is performed.
An unexpected operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3 **Manual numeric command**
When issuing a manual numeric command, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and command have been specified correctly, and that the entered values are valid.
Attempting to operate the machine with an invalid command specified may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

4 **Manual handle feed**
In manual handle feed, rotating the handle with a large scale factor, such as 100, applied causes the tool and table to move rapidly. Careless handling may damage the tool and/or machine, or cause injury to the user.

5 **Disabled override**
If override is disabled (according to the specification in a macro variable) during threading, rigid tapping, or other tapping, the speed cannot be predicted, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

6 **Origin/preset operation**
Basically, never attempt an origin/preset operation when the machine is operating under the control of a program. Otherwise, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the tool, or causing injury to the user.

7 **Workpiece coordinate system shift**
Manual intervention, machine lock, or mirror imaging may shift the workpiece coordinate system. Before attempting to operate the machine under the control of a program, confirm the coordinate system carefully.
If the machine is operated under the control of a program without making allowances for any shift in the workpiece coordinate system, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

⚠ WARNING

8 **Software operator's panel and menu switches**
Using the software operator's panel and menu switches, in combination with the MDI panel, it is possible to specify operations not supported by the machine operator's panel, such as mode change, override value change, and jog feed commands.
Note, however, that if the MDI panel keys are operated inadvertently, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

9 **RESET key**
Pressing the RESET key stops the currently running program. As a result, the servo axes are stopped. However, the RESET key may fail to function for reasons such as an MDI panel problem. So, when the motors must be stopped, use the emergency stop button instead of the RESET key to ensure security.

10 **Manual intervention**
If manual intervention is performed during programmed operation of the machine, the tool path may vary when the machine is restarted. Before restarting the machine after manual intervention, therefore, confirm the settings of the manual absolute switches, parameters, and absolute/incremental command mode.

11 **Feed hold, override, and single block**
The feed hold, feedrate override, and single block functions can be disabled using custom macro system variable #3004. Be careful when operating the machine in this case.

12 **Dry run**
Usually, a dry run is used to confirm the operation of the machine. During a dry run, the machine operates at dry run speed, which differs from the corresponding programmed feedrate. Note that the dry run speed may sometimes be higher than the programmed feed rate.

13 **Cutter and tool nose radius compensation in MDI mode**
Pay careful attention to a tool path specified by a command in MDI mode, because cutter or tool nose radius compensation is not applied. When a command is entered from the MDI to interrupt in automatic operation in cutter or tool nose radius compensation mode, pay particular attention to the tool path when automatic operation is subsequently resumed. Refer to the descriptions of the corresponding functions for details.

14 **Program editing**
If the machine is stopped, after which the machining program is edited (modification, insertion, or deletion), the machine may behave unexpectedly if machining is resumed under the control of that program. Basically, do not modify, insert, or delete commands from a machining program while it is in use.

WARNINGS RELATED TO DAILY MAINTENANCE

⚠ WARNING

1 **Memory backup battery replacement**

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.

When replacing the batteries, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).

Touching the uncovered high-voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The CNC uses batteries to preserve the contents of its memory, because it must retain data such as programs, offsets, and parameters even while external power is not applied.

If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.

When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the contents of the CNC's memory will be lost.

Refer to the Section “Method of replacing battery” in the OPERATOR’S MANUAL (Common to T/M series) for details of the battery replacement procedure.

⚠ WARNING

2 **Absolute pulse coder battery replacement**

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.

When replacing the batteries, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).

Touching the uncovered high-voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The absolute pulse coder uses batteries to preserve its absolute position.

If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.

When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the absolute position data held by the pulse coder will be lost.

Refer to the FANUC SERVO MOTOR αi series Maintenance Manual for details of the battery replacement procedure.

⚠ WARNING

3 **Fuse replacement**

Before replacing a blown fuse, however, it is necessary to locate and remove the cause of the blown fuse.

For this reason, only those personnel who have received approved safety and maintenance training may perform this work.

When replacing a fuse with the cabinet open, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).

Touching an uncovered high-voltage circuit presents an extremely dangerous electric shock hazard.

TABLE OF CONTENTS

SAFETY PRECAUTIONS s-1
DEFINITION OF WARNING, CAUTION, AND NOTE s-1
GENERAL WARNINGS AND CAUTIONS s-2
WARNINGS AND CAUTIONS RELATED TO PROGRAMMING s-3
WARNINGS AND CAUTIONS RELATED TO HANDLING s-5
WARNINGS RELATED TO DAILY MAINTENANCE s-7

I. GENERAL

1 GENERAL 3
1.1 NOTES ON READING THIS MANUAL 6
1.2 NOTES ON VARIOUS KINDS OF DATA 6

II. PROGRAMMING

1 GENERAL 9
1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE-INTERPOLATION 9
1.2 FEED-FEED FUNCTION 11
1.3 PART DRAWING AND TOOL MOVEMENT 12
1.3.1 Reference Position (Machine-specific Position) 12
1.3.2 Coordinate System on Part Drawing and Coordinate System Specified by CNC - Coordinate System 13
1.3.3 How to Indicate Command Dimensions for Moving the Tool (Absolute and Incremental Programming) 18
1.4 CUTTING SPEED - SPINDLE FUNCTION 21
1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING - TOOL FUNCTION 22
1.6 COMMAND FOR MACHINE OPERATIONS - AUXILIARY FUNCTION 23
1.7 PROGRAM CONFIGURATION 24
1.8 TOOL MOVEMENT RANGE - STROKE 26

2 CONTROLLED AXES 27
2.1 NUMBER OF CONTROLLED AXES 27
2.2 NAMES OF AXES 27
2.3 INCREMENT SYSTEM 28
2.4 MAXIMUM STROKE 29

3 PREPARATORY FUNCTION (G FUNCTION) 30
3.1 G CODE LIST IN THE MACHINING CENTER SYSTEM 32
3.2 G CODE LIST IN THE LATHE SYSTEM 36

4 INTERPOLATION FUNCTIONS 40
4.1 POSITIONING (G00) 40
4.2 SINGLE DIRECTION POSITIONING (G60) 41
4.3 LINEAR INTERPOLATION (G01) 44

4.4 CIRCULAR INTERPOLATION (G02, G03) 46
4.5 HELICAL INTERPOLATION (G02, G03) 50
4.6 HELICAL INTERPOLATION B (G02, G03) 51
4.7 SPIRAL INTERPOLATION, CONICAL INTERPOLATION (G02, G03) 52
4.8 POLAR COORDINATE INTERPOLATION (G12.1, G13.1) 58
4.9 CYLINDRICAL INTERPOLATION (G07.1) 66
4.9.1 Cylindrical Interpolation 66
4.9.2 Cylindrical Interpolation by Plane Distance Command 69
4.10 CUTTING POINT INTERPOLATION FOR CYLINDRICAL INTERPOLATION (G07.1) 71
4.11 EXPONENTIAL INTERPOLATION (G02.3, G03.3) 81
4.12 SMOOTH INTERPOLATION (G05.1) 87
4.13 NANO SMOOTHING 92
4.14 NURBS INTERPOLATION (G06.2) 100
4.14.1 NURBS Interpolation Additional Functions 104
4.15 HYPOTHETICAL AXIS INTERPOLATION (G07) 108
4.16 VARIABLE LEAD THREADING (G34) 110
4.17 CIRCULAR THREADING (G35, G36) 111
4.18 SKIP FUNCTION (G31) 115
4.19 MULTI-STEP SKIP (G31) 117
4.20 HIGH-SPEED SKIP SIGNAL (G31) 117
4.21 SKIP POSITION MACRO VARIABLE IMPROVEMENT 118
4.22 CONTINUOUS HIGH-SPEED SKIP FUNCTION 118
4.23 TORQUE LIMIT SKIP 119
4.24 3-DIMENSIONAL CIRCULAR INTERPOLATION 122

5 FEED FUNCTIONS 127
5.1 OVERVIEW 127
5.2 RAPID TRAVERSE 128
5.3 CUTTING FEED 129
5.4 CUTTING FEEDRATE CONTROL 136
5.4.1 Exact Stop (G09, G61), Cutting Mode (G64), Tapping Mode (G63) 137
5.4.2 Automatic Corner Override 137
5.4.2.1 Automatic override for inner corners (G62) 137
5.4.2.2 Internal circular cutting feedrate change 139
5.5 FEEDRATE INSTRUCTION ON IMAGINARY CIRCLE FOR A ROTARY AXIS 140
5.6 DWELL 144

6 REFERENCE POSITION 146
6.1 REFERENCE POSITION RETURN 146
6.2 FLOATING REFERENCE POSITION RETURN (G30.1) 152

7 COORDINATE SYSTEM 154
7.1 MACHINE COORDINATE SYSTEM 154
7.2 WORKPIECE COORDINATE SYSTEM 158
7.2.1 Setting a Workpiece Coordinate System 158
7.2.2 Selecting a Workpiece Coordinate System 160
7.2.3 Changing Workpiece Coordinate System 161

7.2.4 Workpiece Coordinate System Preset (G92.1) 163
7.2.5 Addition of Workpiece Coordinate System Pair (G54.1 or G54) 165
7.2.6 Automatic Coordinate System Setting 167
7.2.7 Workpiece Coordinate System Shift 167
7.3 LOCAL COORDINATE SYSTEM 169
7.4 PLANE SELECTION 170
7.5 PLANE CONVERSION FUNCTION 171

8 COORDINATE VALUE AND DIMENSION 177
8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING 177
8.2 INCH/METRIC CONVERSION (G20, G21) 179
8.3 DECIMAL POINT PROGRAMMING 182
8.4 DIAMETER AND RADIUS PROGRAMMING 183
8.5 DIAMETER AND RADIUS SETTING SWITCHING FUNCTION 184

9 SPINDLE SPEED FUNCTION (S FUNCTION) 187
9.1 SPECIFYING THE SPINDLE SPEED WITH A CODE 187
9.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5-DIGIT COMMAND) 187
9.3 CONSTANT SURFACE SPEED CONTROL (G96, G97) 187
9.4 SPINDLE POSITIONING FUNCTION 191
9.4.1 Spindle Orientation 192
9.4.2 Spindle Positioning 193
9.4.3 Canceling Spindle Positioning 194
9.5 SPINDLE SPEED FLUCTUATION DETECTION 196
9.6 SPINDLE CONTROL WITH SERVO MOTOR 199
9.6.1 Spindle Control with Servo Motor 199
9.6.2 Spindle Indexing Function 200

10 TOOL FUNCTION (T FUNCTION) 202
10.1 TOOL SELECTION FUNCTION 202
10.2 TOOL MANAGEMENT FUNCTION 203
10.3 TOOL MANAGEMENT EXTENSION FUNCTION 217
10.3.1 Customization of Tool Management Data Display 217
10.3.2 Setting of Spindle Position / Standby Position Display 221
10.3.3 Input of Customize Data with the Decimal Point 223
10.3.4 Protection of Various Tool Information Items with the KEY Signal 225
10.3.5 Selection of a Tool Life Count Period 225
10.3.6 Each tool Data Screen 226
10.3.7 Total Life Time Display for Tools of The Same Type 226
10.4 TOOL MANAGEMENT FUNCTION FOR OVERSIZE TOOLS 226
10.5 TOOL LIFE MANAGEMENT 228
10.5.1 Tool Life Management Data 229
10.5.2 Registering, Changing, and Deleting Tool Life Management Data 231
10.5.3 Tool Life Management Commands in Machining Program 236
10.5.4 Tool Life Counting and Tool Selection 242
10.5.5 Tool Life Count Restart M Code 244
10.5.6 Disabling Life Count 246
10.5.7 Remaining Tool Number Check Function 246

11 AUXILIARY FUNCTION 248

11.1 AUXILIARY FUNCTION (M FUNCTION) 248
11.2 MULTIPLE M COMMANDS IN A SINGLE BLOCK 249
11.3 M CODE GROUPING FUNCTION 250
11.3.1 Setting an M Code Group Number Using the Setting Screen 250
11.3.2 Setting an M Code Group Number Using a Program 251
11.3.3 M Code Group Check Function 252
11.4 SECOND AUXILIARY FUNCTIONS (B CODES) 252

12 PROGRAM MANAGEMENT 255
12.1 FOLDERS 255
12.1.1 Folder Configuration 255
12.1.2 Folder Attributes 257
12.1.3 Default Folders 257
12.2 PROGRAMS 258
12.2.1 Program Name 258
12.2.2 Program Attributes 260
12.3 RELATION WITH CONVENTIONAL FUNCTIONS 260
12.3.1 Relation with Folders 260
12.3.2 Relation with Program Names 261
12.3.3 Related Parameters 263
12.3.4 Part Program Storage Size / Number of Registerable Programs 263

13 PROGRAM CONFIGURATION 265
13.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS 266
13.2 PROGRAM SECTION CONFIGURATION 268
13.3 SUBPROGRAM (M98, M99) 274

14 FUNCTIONS TO SIMPLIFY PROGRAMMING 279
14.1 FIGURE COPYING (G72.1, G72.2) 279
14.2 3-DIMENSIONAL COORDINATE SYSTEM CONVERSION 284

15 COMPENSATION FUNCTION 295
15.1 TOOL LENGTH COMPENSATION (G43, G44, G49) 295
15.1.1 Overview 295
15.1.2 G53, G28, G30, and G30.1 Commands in Tool Length Compensation Mode 300
15.2 SCALING (G50, G51) 302
15.3 PROGRAMMABLE MIRROR IMAGE (G50.1, G51.1) 309
15.4 NORMAL DIRECTION CONTROL (G40.1,G41.1,G42.1) 311
15.5 WORKPIECE SETTING ERROR COMPENSATION 315
15.6 TOOL OFFSET FOR MILLING AND TURNING FUNCTION (G43.7) 361
15.7 TOOL OFFSET CONVERSION FUNCTION (G44.1) 367

16 CUSTOM MACRO 380
16.1 VARIABLES 380
16.2 SYSTEM VARIABLES 385
16.3 READING AND WRITING VARIABLES FOR ANOTHER PATH 436
16.4 ARITHMETIC AND LOGIC OPERATION 440
16.5 INDIRECT AXIS ADDRESS SPECIFICATION 447
16.6 READING PARAMETERS 448
16.7 MACRO STATEMENTS AND NC STATEMENTS 449

16.8 BRANCH AND REPETITION 450
16.8.1 Unconditional Branch (GOTO Statement) 450
16.8.2 GOTO Statement Using Stored Sequence Numbers 450
16.8.3 Conditional Branch (IF Statement) 451
16.8.4 Repetition (WHILE Statement) 452
16.8.5 Precision Specification for Macro Relational Operators 454
16.9 MACRO CALL 456
16.9.1 Simple Call (G65) 457
16.9.2 Modal Call: Call After the Move Command (G66) 464
16.9.3 Modal Call: Each Block Call (G66.1) 467
16.9.4 Macro Call Using a G Code 469
16.9.5 Macro Call Using a G Code (Specification of Multiple Definitions) 470
16.9.6 Macro Call Using a G Code with a Decimal Point (Specification of Multiple Definitions) 471
16.9.7 Macro Call Using an M Code 471
16.9.8 Macro Call Using an M Code (Specification of Multiple Definitions) 473
16.9.9 Special Macro Call using M code 474
16.9.10 Subprogram Call Using an M Code 476
16.9.11 Subprogram Call Using an M Code (Specification of Multiple Definitions) 477
16.9.12 Subprogram Calls Using a T Code 477
16.9.13 Subprogram Calls Using an S Code 478
16.9.14 Subprogram Calls Using a Secondary Auxiliary Function 478
16.9.15 Subprogram Call Using a Specific Address 479
16.10 MACRO CALL ARGUMENT FOR AXIS NAME EXPANSION 482
16.11 PROCESSING MACRO STATEMENTS 483
16.12 REGISTERING CUSTOM MACRO PROGRAMS 486
16.13 CODES AND RESERVED WORDS USED IN CUSTOM MACROS 486
16.14 EXTERNAL OUTPUT COMMANDS 486
16.15 RESTRICTIONS 490
16.16 INTERRUPTION TYPE CUSTOM MACRO 492
16.16.1 Specification Method 493
16.16.2 Details of Functions 494

17 REAL-TIME CUSTOM MACRO 501
17.1 TYPES OF REAL TIME MACRO COMMANDS 503
17.1.1 Modal Real Time Macro Command / One-shot Real Time Macro Command 503
17.2 VARIABLES 508
17.2.1 Variables Dedicated to Real Time Custom Macros 509
17.2.1.1 System variables 509
17.2.1.2 Real time macro variables (RTM variables) 511
17.2.2 Custom Macro Variables 512
17.2.2.1 System variables 513
17.2.2.2 Local variables 514
17.3 ARITHMETIC AND LOGICAL OPERATION 514
17.4 CONTROL ON REAL TIME MACRO COMMANDS 515
17.4.1 Conditional Branch (ZONCE Statement) 516
17.4.2 Condition Transition (ZEDGE Statement) 517
17.4.3 Repetition (ZWHILE Statement) 518
17.4.4 Multi-statement (ZDO...ZEND Statement) 519
17.5 MACRO CALL 521
17.6 OTHERS 522
17.7 AXIS CONTROL COMMAND 523

17.8 NOTES 534
17.9 LIMITATION 535

18 PROGRAMMABLE PARAMETER INPUT (G10) 538

19 PATTERN DATA INPUT 542
19.1 OVERVIEW 542
19.2 EXPLANATION 542
19.3 EXPLANATION OF OPERATION 546
19.4 DEFINITION OF THE SCREEN 550
19.4.1 Definition of the Pattern Menu Screen 551
19.4.2 Definition of the Custom Macro Screen 553
19.4.3 Setting the Character-codes 556

20 HIGH-SPEED CUTTING FUNCTIONS 561
20.1 AI CONTOUR CONTROL FUNCTION I AND AI CONTOUR CONTROL FUNCTION II (G05.1) 561
20.2 MACHINING CONDITION SELECTING FUNCTION 575
20.3 MACHINING QUALITY LEVEL ADJUSTMENT 575
20.4 HIGH-SPEED CYCLE MACHINING 577
20.5 HIGH-SPEED BINARY PROGRAM OPERATION 578
20.6 OPTIMUM ACCELERATION/DECELERATION FOR RIGID TAPPING 579
20.7 PATH TABLE OPERATION 581

21 AXIS CONTROL FUNCTIONS 607
21.1 AXIS SYNCHRONOUS CONTROL 607
21.2 POLYGON TURNING (G50.2, G51.2) 609
21.3 SYNCHRONOUS, COMPOSITE AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND (G50.4, G51.4, G50.5, G51.5, G50.6, AND G51.6) 614
21.4 ROTARY AXIS ROLL-OVER 617
21.4.1 Rotary Axis Roll-over 617
21.4.2 Rotary Axis Control 618
21.5 TOOL RETRACT AND RECOVER 619
21.5.1 Improvement of Tool compensation for Tool Retract and Recover 622
21.6 ELECTRONIC GEAR BOX 626
21.6.1 Electronic Gear Box 626
21.6.2 Spindle Electronic Gear Box 632
21.6.3 Electronic Gear Box Automatic Phase Synchronization 639
21.6.4 Skip Function for EGB Axis 643
21.6.5 Electronic Gear Box 2 Pair 645
21.6.5.1 Specification method (G80.5, G81.5) 645
21.6.5.2 Description of commands compatible with those for a hobbing machine (G80, G81) 648
21.6.5.3 Controlled axis configuration example 651
21.6.5.4 Sample programs 652
21.6.5.5 Synchronization ratio specification range 655
21.6.5.6 Retract function 658
21.6.6 U-axis Control 658
21.6.7 U-axis Control 2 Pairs 660
21.7 TANDEM CONTROL 661

21.8 PIVOT AXIS CONTROL 661
21.9 CHOPPING FUNCTION 664
21.10 SKIP FUNCTION FOR FLEXIBLE SYNCHRONOUS CONTROL 671
21.11 CHOPPING FUNCTION BY FLEXIBLE SYNCHRONOUS CONTROL 673
21.12 HIGH PRECISION OSCILLATION FUNCTION 674

22 5-AXIS MACHINING FUNCTION 682
22.1 TOOL CENTER POINT CONTROL 682
22.2 HIGH-SPEED SMOOTH TCP 739
22.2.1 High-speed Smooth TCP 739
22.2.1.1 Rotation axes compensation (G43.4L1, G43.5L1) 744
22.2.1.2 Smooth control (G43.4P3, G43.5P3) 749
22.2.2 Tolerance change in High-speed Smooth TCP mode 755
22.2.2.1 Tolerance change in rotation axes compensation (G43.4L1, G43.5L1) 755
22.2.2.2 Tolerance change in smooth control (G43.4P3, G43.5P3) 758
22.2.3 Information Display in High-speed Smooth TCP 758
22.3 EXPANSION OF AXIS MOVE COMMAND IN TOOL CENTER POINT CONTROL 764
22.4 TOOL POSTURE CONTROL 766
22.5 CUTTING POINT COMMAND 776
22.6 TILTED WORKING PLANE INDEXING 786
22.6.1 Tilted Working Plane Indexing 786
22.6.1.1 Tilted working plane indexing based on Eulerian angle 789
22.6.1.2 General specifications of the tilted working plane indexing 790
22.6.1.3 Tilted working plane indexing based on roll-pitch-yaw 793
22.6.1.4 Tilted working plane indexing based on three points 795
22.6.1.5 Tilted working plane indexing based on two vectors 799
22.6.1.6 Tilted working plane indexing based on projection angles 802
22.6.1.7 Tilted working plane indexing by tool axis direction 805
22.6.2 Multiple command of tilted working plane indexing 815
22.6.2.1 Absolute multiple command 815
22.6.2.2 Incremental multiple command 817
22.6.3 Tool Axis Direction Control 819
22.6.3.1 Tool axis direction control 819
22.6.3.2 Tool center point retention type tool axis direction control 836
22.6.4 Tilted Working Plane Indexing in Tool Length Compensation 840
22.7 INCLINED ROTARY AXIS CONTROL 847
22.8 3-DIMENSIONAL CUTTER COMPENSATION 850
22.8.1 Cutter Compensation in Tool Rotation Type Machine 852
22.8.1.1 Tool side offset 853
22.8.1.2 Leading edge offset 869
22.8.1.3 Tool tip position (cutting point) command 873
22.8.2 Cutter Compensation in Table Rotation Type Machine 876
22.8.3 Cutter Compensation in Composite Type Machine 881
22.8.4 Interference Check and Interference Avoidance 886
22.8.5 Restrictions 889
22.8.5.1 Restrictions common to machine configurations 889
22.8.5.2 Restriction on tool rotation type 891
22.8.5.3 Restriction on machine configurations having table rotation axes (table rotation type and composite type) 892
22.8.6 Examples 895
22.9 EXPANSION OF THE WAY TO SET 5-AXIS MACHINING FUNCTION PARAMETERS 900

22.10 MACHINE CONFIGURATION SELECTING FUNCTION 903
22.10.1 Machine Configuration Selecting Screen 903
22.10.2 Switching Machine Configuration 904
22.10.3 Setting Machine Configuration Data 906
22.10.4 Inputting and Outputting Machine Configuration Data 909

23 MUITI-PATH CONTROL FUNCTION 911
23.1 OVERVIEW 911
23.2 WAITING FUNCTION FOR PATHS 912
23.3 COMMON MEMORY BETWEEN EACH PATH 917
23.4 SPINDLE CONTROL BETWEEN EACH PATH 918
23.5 SYNCHRONOUS/COMPOSITE/SUPERIMPOSED CONTROL 919

III. OPERATION

1 GENERAL 925
1.1 MANUAL OPERATION 925
1.2 TOOL MOVEMENT BY PROGRAMING - AUTOMATIC OPERATION 926
1.3 AUTOMATIC OPERATION 927
1.4 TESTING A PROGRAM 928
1.4.1 Check by Running the Machine 928
1.4.2 How to View the Position Display Change without Running the Machine 930
1.5 EDITING A PROGRAM 930
1.6 DISPLAYING AND SETTING DATA 930
1.7 DISPLAY 933
1.7.1 Program Display 933
1.7.2 Current Position Display 934
1.7.3 Alarm Display 935
1.7.4 Parts Count Display, Run Time Display 936

2 OPERATIONAL DEVICES 937
2.1 POWER ON/OFF 937
2.1.1 Turning on the Power 937
2.1.2 Power Disconnection 938
2.2 SETTING AND DISPLAY UNITS 939
2.2.1 8.4" LCD CNC Display Panel 939
2.2.2 10.4" LCD CNC Display Panel (12.1"/15"/19" LCD CNC Display Panel) 940
2.2.3 Standard MDI Unit (ONG Key) 941
2.2.4 Standard MDI Unit (QWERTY Key) 942
2.2.5 Small MDI Unit (ONG Key) 943
2.3 EXPLANATION OF THE MDI UNIT 944
2.4 FUNCTION KEYS AND SOFT KEYS 947
2.4.1 General Screen Operations 947
2.4.2 Function Keys 948
2.4.3 Soft Keys 949
2.5 EXTERNAL I/O DEVICES 957

3 MANUAL OPERATION 959
3.1 MANUAL REFERENCE POSITION RETURN 959
3.2 JOG FEED (JOG) 960
3.3 INCREMENTAL FEED 962

3.4 MANUAL HANDLE FEED 963
3.5 MANUAL ABSOLUTE ON AND OFF 966
3.6 MANUAL LINEAR/CIRCULAR INTERPOLATION 970
3.7 RIGID TAPPING BY MANUAL HANDLE 974
3.8 MANUAL NUMERICAL COMMAND 976
3.9 3-DIMENSIONAL MANUAL FEED 984
3.9.1 Tool Axis Direction Handle Feed / Tool Axis Direction JOG Feed / Tool Axis Direction Incremental Feed 985
3.9.2 Tool Axis Right-Angle Direction Handle Feed / Tool Axis Right-Angle Direction JOG Feed / Tool Axis Right-Angle Direction Incremental Feed 987
3.9.3 Tool Tip Center Rotation Handle Feed / Tool Tip Center Rotation JOG Feed / Tool Tip Center Rotation Incremental Feed 990
3.9.4 Table Vertical Direction Handle Feed / Table Vertical Direction JOG Feed / Table Vertical Direction Incremental Feed 992
3.9.5 Table Horizontal Direction Handle Feed / Table Horizontal Direction JOG Feed / Table Horizontal Direction Incremental Feed 994
3.10 DISTANCE CODED LINEAR SCALE INTERFACE 997
3.10.1 Procedure for Reference Position Establishment 998
3.10.2 Reference Position Return 999
3.10.3 Distance Coded Rotary Encoder 999
3.10.4 Axis Synchronization Control 999
3.10.5 Axis Control by PMC 1001
3.10.6 Angular Axis Control 1001
3.10.7 Note 1001
3.11 LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL) 1002

4 AUTOMATIC OPERATION 1007
4.1 MEMORY OPERATION 1007
4.2 MDI OPERATION 1009
4.3 DNC OPERATION 1012
4.4 SCHEDULE OPERATION 1015
4.5 EXTERNAL SUBPROGRAM CALL (M198) 1018
4.6 EXTERNAL SUBPROGRAM CALLS USING THE DATA SERVER AVAILABLE IN MULTI-PATH SYSTEMS 1022
4.7 MANUAL HANDLE INTERRUPTION 1023
4.7.1 Manual Interruption of 3-dimensional Coordinate System Conversion 1029
4.8 MIRROR IMAGE 1030
4.9 PROGRAM RESTART 1031
4.9.1 Auxiliary Function Output in Program Restart Function 1047
4.10 QUICK PROGRAM RESTART 1051
4.10.1 Suppress Motion of Quick Program Restart 1068
4.10.2 Quick Program Restart for a Machining Cycle 1072
4.11 TOOL RETRACT AND RECOVER 1080
4.11.1 Retract 1082
4.11.2 Withdrawal 1083
4.11.3 Return 1083
4.11.4 Repositioning 1084
4.11.5 Tool Retract and Recover for Threading 1084
4.11.6 Operation Procedure for a Canned Cycle for Drilling 1087
4.12 MANUAL INTERVENTION AND RETURN 1089

4.13 RETRACE 1092
4.14 ACTIVE BLOCK CANCEL FUNCTION 1101

5 TEST OPERATION 1105
5.1 MACHINE LOCK AND AUXILIARY FUNCTION LOCK 1105
5.2 FEEDRATE OVERRIDE 1106
5.3 RAPID TRAVERSE OVERRIDE 1107
5.4 DWELL/AUXILIARY FUNCTION TIME OVERRIDE 1107
5.5 DRY RUN 1109
5.6 SINGLE BLOCK 1110
5.7 HIGH SPEED PROGRAM CHECK FUNCTION 1111
5.8 MANUAL HANDLE RETRACE 1113
5.8.1 Displaying Program Being Executed in Backward Movement 1124
5.9 AUXILIARY FUNCTION OUTPUT BLOCK REVERSE MOVEMENT FOR MANUAL HANDLE RETRACE 1126
5.10 MANUAL HANDLE RETRACE FUNCTION FOR MULTI-PATH 1126
5.11 EXTENSION OF THE MANUAL HANDLE RETRACE FUNCTION 1130

6 SAFETY FUNCTIONS 1134
6.1 EMERGENCY STOP 1134
6.2 OVERTRAVEL 1134
6.3 STORED STROKE CHECK 1135
6.4 STROKE LIMIT CHECK BEFORE MOVE 1139
6.5 STROKE LIMIT AREA CHANGING FUNCTION 1141
6.6 WRONG OPERATION PREVENTION FUNCTIONS 1142
6.6.1 Functions that are Used When Data is Set 1142
6.6.1.1 Input data range check 1143
6.6.1.2 Confirmation of incremental input 1144
6.6.1.3 Prohibition of the absolute input by the soft key 1145
6.6.1.4 Confirmation of the deletion of the program 1145
6.6.1.5 Confirmation of the deletion of all data 1146
6.6.1.6 Confirmation of a data update during the data setting process 1146
6.6.2 Functions that are Used when the Program is Executed 1147
6.6.2.1 Display of updated modal information 1147
6.6.2.2 Start check signal 1147
6.6.2.3 Axis status display 1148
6.6.2.4 Confirmation of the start from a middle block 1149
6.6.2.5 Data range check 1150
6.6.2.6 Maximum incremental value check 1151
6.6.3 Setting Screen 1151
6.6.3.1 Operation confirmation function setting screen 1151
6.6.3.2 Tool offset range setting screen 1153
6.6.3.3 Workpiece origin offset range setting screen 1156
6.6.3.4 Y-axis tool offset range setting screen 1158
6.6.3.5 Workpiece shift range setting screen 1159

7 ALARM AND SELF-DIAGNOSIS FUNCTIONS 1161
7.1 ALARM DISPLAY 1161
7.2 ALARM HISTORY DISPLAY 1163
7.3 CHECKING BY DIAGNOSTIC DISPLAY 1164
7.4 RETURN FROM THE ALARM SCREEN 1166
7.4.1 Return from the Alarm Screen 1166

7.4.2 Relationship with Other Functions....1167

8 DATA INPUT/OUTPUT....1169

8.1 OVERWRITING FILES ON A MEMORY CARD/USB MEMORY....1171

8.2 INPUT/OUTPUT ON EACH SCREEN....1173

8.2.1 Inputting and Outputting a Program....1174
8.2.1.1 Inputting a program....1174
8.2.1.2 Outputting a program....1176
8.2.1.3 Inputting and outputting of O8-digit....1177
8.2.2 Inputting and Outputting Parameters....1178
8.2.2.1 Inputting parameters....1178
8.2.2.2 Outputting parameters....1179
8.2.3 Inputting and Outputting Offset Data....1180
8.2.3.1 Inputting offset data....1180
8.2.3.2 Outputting offset data....1181
8.2.4 Inputting and Outputting Pitch Error Compensation Data....1185
8.2.4.1 Inputting pitch error compensation data....1185
8.2.4.2 Outputting pitch error compensation data....1186
8.2.4.3 Input/output format of pitch error compensation data....1187
8.2.5 Inputting and Outputting 3-dimensional Error Compensation Data....1188
8.2.5.1 Inputting 3-dimensional error compensation data....1188
8.2.5.2 Outputting 3-dimensional error compensation data....1190
8.2.5.3 Input/output format of 3-dimensional error compensation data....1190
8.2.6 Inputting and Outputting Three-dimensional Rotary Error Compensation Data....1192
8.2.6.1 Inputting three-dimensional rotary error compensation data....1192
8.2.6.2 Outputting three-dimensional rotary error compensation data....1193
8.2.6.3 Input/output format of three-dimensional rotary error compensation data....1194
8.2.7 Inputting and Outputting Custom Macro Common Variables....1196
8.2.7.1 Inputting custom macro common variables....1196
8.2.7.2 Outputting custom macro common variables....1197
8.2.8 Inputting and Outputting Workpiece Coordinates System Data....1199
8.2.8.1 Inputting workpiece coordinate system data....1199
8.2.8.2 Outputting workpiece coordinate system data....1200
8.2.9 Inputting and Outputting Operation History Data....1200
8.2.9.1 Outputting operation history data....1201
8.2.9.2 Inputting operation history signal selection data....1201
8.2.9.3 Outputting operation history signal section data....1202
8.2.9.4 Input/output format of operation history signal selection data....1203
8.2.10 Inputting and Outputting Tool Management Data....1204
8.2.10.1 Inputting tool management data....1204
8.2.10.2 Outputting tool management data....1205
8.2.10.3 Inputting magazine data....1206
8.2.10.4 Outputting magazine data....1208
8.2.10.5 Inputting tool life status name data....1208
8.2.10.6 Outputting tool life status name data....1209
8.2.10.7 Inputting name data of customize data....1210
8.2.10.8 Outputting name data of customize data....1211
8.2.10.9 Inputting customize data displayed as tool management data....1211
8.2.10.10 Outputting customize data displayed as tool management data....1212
8.2.10.11 Inputting spindle waiting position name data....1213
8.2.10.12 Outputting spindle waiting position name data....1214
8.2.10.13 Inputting decimal point position data of customize data....1215
8.2.10.14 Outputting decimal point position data of customize data....1216
8.2.10.15 Inputting tool geometry data....1217
8.2.10.16 Outputting tool geometry data....1218
8.2.11 Inputting and Outputting Workpiece Setting Error Compensation Value....1219
8.2.11.1 Inputting values on the workpiece setting error compensation screen....1219

8.2.11.2 Outputting values on the workpiece setting error compensation screen 1220
8.2.11.3 Input/output format of workpiece setting error values 1221
8.2.12 Inputting and Outputting Tool Life Management Data 1222
8.2.12.1 Inputting tool life management data 1222
8.2.12.2 Outputting tool life management data 1223
8.3 INPUT/OUTPUT ON THE ALL IO SCREEN 1224
8.3.1 Inputting/Outputting a Program 1227
8.3.2 Inputting/Outputting all Programs and Folders 1229
8.3.3 Inputting and Outputting Parameters 1231
8.3.4 Inputting and Outputting Offset Data 1232
8.3.5 Inputting/Outputting Pitch Error Compensation Data 1234
8.3.6 Inputting/Outputting Custom Macro Common Variables 1235
8.3.7 Inputting and Outputting Workpiece Coordinates System Data 1237
8.3.8 Inputting and Outputting Operation History Data 1238
8.3.9 Inputting and Outputting Tool Management Data 1239
8.3.10 Inputting and Outputting All Tool Management Data at a Time 1244
8.3.11 Inputting and Outputting Workpiece Setting Error Compensation Value 1247
8.3.12 File Format and Error Messages 1248
8.4 EMBEDDED ETHERNET OPERATIONS 1249
8.4.1 FTP File Transfer Function 1249
8.5 SCREEN HARD COPY FUNCTION 1255
8.6 USB FUNCTION 1256

9 CREATING PROGRAMS 1258
9.1 CREATING PROGRAMS USING THE MDI PANEL 1258
9.2 AUTOMATIC INSERTION OF SEQUENCE NUMBERS 1259
9.3 CREATING PROGRAMS IN TEACH IN MODE (PLAYBACK) 1260

10 EDITING PROGRAMS 1263
10.1 EDIT DISABLE ATTRIBUTE AND DISPLAY DISABLE ATTRIBUTE 1263
10.2 INSERTING, ALTERING AND DELETING A WORD 1264
10.2.1 Word Search 1265
10.2.2 Heading a Program 1267
10.2.3 Inserting a Word 1267
10.2.4 Altering a Word 1268
10.2.5 Deleting a Word 1268
10.3 REPLACING A WORD OR ADDRESS 1269
10.4 DELETING BLOCKS 1271
10.4.1 Deleting a Block 1271
10.4.2 Deleting Multiple Blocks 1271
10.5 PROGRAM SEARCH 1272
10.6 SEQUENCE NUMBER SEARCH 1273
10.7 DELETING PROGRAMS 1275
10.7.1 Deleting One Program 1275
10.7.2 Deleting All Programs 1275
10.8 EDITING OF CUSTOM MACROS 1275
10.9 CURSOR MOVEMENT LIMITATIONS ON PROGRAM EDITING 1276
10.10 PASSWORD FUNCTION 1277
10.11 EDITING PROGRAM CHARACTERS 1279
10.11.1 Available Keys 1281
10.11.2 Input Mode 1282
10.11.3 Line Number Display 1282

10.11.4 Search 1282
10.11.5 Replacement 1283
10.11.6 Reversing Edit Operations (Undo Function) 1284
10.11.7 Copy 1284
10.11.8 Cut 1284
10.11.9 Paste 1284
10.11.10 Saving 1285
10.11.11 Creation 1285
10.11.12 Line Search 1285
10.12 PROGRAM COPY FUNCTION 1286
10.12.1 Copying and Moving Files between Devices 1287
10.13 KEYS AND PROGRAM ENCRYPTION 1290
10.14 SIMULTANEOUS EDITING OF MULTIPATH PROGRAMS 1293
10.15 MULTI-PATH EDITING FUNCTION 1296
10.15.1 Overview 1296
10.15.2 Details 1296

11 PROGRAM MANAGEMENT 1301
11.1 SELECTING A DEVICE 1302
11.1.1 Selecting a Memory Card Program as a Device 1302
11.1.2 Selecting a From Cassette as a Device 1306
11.1.3 Selecting a USB Memory as a Device 1307
11.2 CREATING A FOLDER 1308
11.3 RENAMING A FOLDER 1309
11.4 CHANGING CURRENT FOLDER 1309
11.5 CHANGING FOLDER ATTRIBUTES 1311
11.6 DELETING A FOLDER 1311
11.7 SELECTING A DEFAULT FOLDER 1312
11.8 RENAMING A FILE 1313
11.9 DELETING A FILE 1313
11.10 CHANGING FILE ATTRIBUTES 1314
11.11 SELECTING A MAIN PROGRAM 1315
11.12 PROGRAM AND FOLDER COPY/MOVE FUNCTION 1315
11.12.1 Copy and movement between different devices 1318
11.13 FOLDER MANAGEMENT 1319
11.13.1 Program Management under the Path Folder 1321
11.13.2 Program Management only in the Path Folder 1322
11.13.3 Folder for Subprogram/Macro Calls 1324
11.14 PROGRAM VERIFICATION 1324

12 SETTING AND DISPLAYING DATA 1326
12.1 SCREENS DISPLAYED BY FUNCTION KEY 

....1344
12.1.1 Position Display in the Workpiece Coordinate System 1344
12.1.2 Position Display in the Relative Coordinate System 1346
12.1.3 Overall Position Display 1349
12.1.4 Workpiece Coordinate System Preset 1350
12.1.5 Actual Feedrate Display 1351
12.1.6 Display of Run Time and Parts Count 1353
12.1.7 Setting the Floating Reference Position 1355
12.1.8 Operating Monitor Display 1355

12.1.9 Display of 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount)....1357
12.1.10 Overall Position Display (15/19-inch Display Unit)....1360
12.1.11 Workpiece Coordinate System Preset (15/19-inch Display Unit)....1363
12.1.12 Actual Feedrate Display (15/19-inch Display Unit)....1363
12.1.13 Display of Run Time and Parts Count (15/19-inch Display Unit)....1365
12.1.14 Setting the Floating Reference Position (15/19-inch Display Unit)....1367
12.1.15 Operating Monitor Display (15/19-inch Display Unit)....1367
12.1.16 Display of 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount) (15/19-inch Display Unit)....1369

12.2 SCREENS DISPLAYED BY FUNCTION KEY  1372

12.2.1 Program Contents Display....1372
12.2.1.1 Displaying the executed block....1373
12.2.1.2 Text display....1375
12.2.2 Editing a Program....1376
12.2.3 Program Screen for MDI Operation....1377
12.2.4 Program Folder Screen....1378
12.2.4.1 Split display on the program folder screen....1379
12.2.4.2 Folder tree display....1383
12.2.5 Next Block Display Screen....1385
12.2.6 Program Check Screen....1386
12.2.7 Background Editing....1386
12.2.8 Stamping the Machining Time....1392
12.2.9 Screen for Assistance in Entering Tilted Working Plane Indexing....1399
12.2.9.1 Command type selection screen....1405
12.2.9.2 Tilted working plane data setting screen....1406
12.2.9.3 Details of the tilted working plane data setting screen....1409
12.2.9.4 Limitation....1416
12.2.10 Program Contents Display (15/19-inch Display Unit)....1416
12.2.10.1 Displaying the executed block....1417
12.2.11 Editing a Program (15/19-inch Display Unit)....1417
12.2.12 Program Screen for MDI Operation (15/19-inch Display Unit)....1419
12.2.13 Program Folder Screen (15/19-inch Display Unit)....1419
12.2.13.1 Split display on the program folder screen....1420
12.2.13.2 Folder tree display (15/19-inch display unit)....1424
12.2.14 Next Block Display Screen (15/19-inch Display Unit)....1424
12.2.15 Program Check Screen (15/19-inch Display Unit)....1425
12.2.16 Background Editing (15/19-inch Display Unit)....1426
12.2.17 Stamping the Machining Time (15/19-inch Display Unit)....1431
12.2.18 Screen for Assistance in Entering Tilted Working Plane Indexing (15/19-inch Display Unit)....1439
12.2.18.1 Command type selection screen....1445
12.2.18.2 Tilted working plane data setting screen....1446
12.2.18.3 Details of the tilted working plane data setting screen....1449
12.2.18.4 Limitation....1456

12.3 SCREENS DISPLAYED BY FUNCTION KEY  1457

12.3.1 Displaying and Entering Setting Data....1458
12.3.2 Sequence Number Comparison and Stop....1461
12.3.3 Displaying and Setting Run Time, Parts Count, and Time....1462
12.3.4 Displaying and Setting the Workpiece Origin Offset Value....1464
12.3.5 Direct Input of Workpiece Origin Offset Value Measured....1465
12.3.6 Displaying and Setting Custom Macro Common Variables....1466
12.3.7 Displaying and Setting Real Time Custom Macro Data....1468
12.3.8 Displaying and Setting the Software Operator's Panel....1469

12.3.9 Setting and Displaying Tool Management Data 1472
12.3.9.1 Displaying and setting magazine screen 1472
12.3.9.2 Displaying and setting tool management screen 1473
12.3.9.3 Each tool data screen 1478
12.3.9.4 Displaying the total life of tools of the same type....1481
12.3.9.5 Tool geometry data screen 1484
12.3.10 Displaying and Switching the Display Language 1488
12.3.11 Protection of Data at Eight Levels....1490
12.3.11.1 Operation level setting 1490
12.3.11.2 Password modification 1491
12.3.11.3 Protection level setting....1493
12.3.11.4 Setting the change protection level and output protection level of a program 1495
12.3.12 Precision Level Selection 1496
12.3.13 Machining Level Selection....1498
12.3.13.1 Smoothing level selection 1498
12.3.13.2 Precision level selection....1499
12.3.14 Machining Quality Level Selection....1499
12.3.15 Displaying and Setting Tool Life Management Data....1500
12.3.15.1 Tool life management (list screen)....1502
12.3.15.2 Tool life management (group editing screen) 1505
12.3.16 Displaying and Setting Workpiece Setting Error Compensation Data....1511
12.3.17 Displaying and Setting Pattern Data Inputs....1512
12.3.18 Setting and Displaying Data When the Tool Offset for Milling and Turning Function Is Enabled....1514
12.3.19 Displaying and Entering Setting Data (15/19-inch Display Unit)....1517
12.3.20 Sequence Number Comparison and Stop (15/19-inch Display Unit)....1519
12.3.21 Displaying and Setting Run Time, Parts Count, and Time (15/19-inch Display Unit) 1521
12.3.22 Displaying and Setting the Workpiece Origin Offset Value (15/19-inch Display Unit) 1523
12.3.23 Direct Input of Workpiece Origin Offset Value Measured (15/19-inch Display Unit) 1524
12.3.24 Displaying and Setting Custom Macro Common Variables (15/19-inch Display Unit) 1525
12.3.25 Displaying and Setting Real Time Custom Macro Data (15/19-inch Display Unit) 1527
12.3.26 Displaying and Setting the Software Operator’s Panel (15/19-inch Display Unit) 1528
12.3.27 Setting and Displaying Tool Management Data (15/19-inch Display Unit) 1531
12.3.27.1 Displaying and setting magazine screen (15-inch display unit)....1531
12.3.27.2 Displaying and setting tool management screen (15-inch display unit) 1532
12.3.27.3 Each tool data screen (15-inch display unit)....1538
12.3.27.4 Displaying the total life of tools of the same type (15-inch display unit)....1540
12.3.27.5 Tool geometry data screen (15-inch display unit)....1544
12.3.28 Displaying and Switching the Display Language (15/19-inch Display Unit)....1549
12.3.29 Protection of Data at Eight Levels (15/19-inch Display Unit) 1550
12.3.29.1 Operation level setting (15-inch display unit)....1550
12.3.29.2 Password modification (15-inch display unit) 1552
12.3.29.3 Protection level setting (15-inch display unit) 1553
12.3.29.4 Setting the change protection level and output protection level of a program (15-inch display unit)....1555
12.3.30 Precision Level Selection (15/19-inch Display Unit)....1556
12.3.31 Machining Level Selection (15/19-inch Display Unit) 1558
12.3.31.1 Smoothing level selection 1558
12.3.31.2 Precision level selection....1559

12.3.32 Machining Quality Level Selection (15/19-inch Display Unit) 1559
12.3.33 Displaying and Setting Tool Life Management Data (15/19-inch Display Unit)1561
12.3.33.1 Tool life management (list screen) (15-inch display unit) 1562
12.3.33.2 Tool life management (group editing screen) (15-inch display unit).....1565
12.3.34 Displaying and Setting Workpiece Setting Error Compensation Data (15/19-inch Display Unit) 1572
12.3.35 Displaying and Setting Pattern Data Inputs (15/19-inch Display Unit) 1573
12.3.36 Built-in 3D Interference Check 1576
12.3.36.1 Monitor menu screen 1576
12.3.36.2 Tool monitor screen 1579
12.3.36.3 Tool holder and object monitor screen 1580
12.3.36.4 Figure setting menu screen 1581
12.3.36.5 Object figure setting screen.....1582
12.3.36.6 Tool holder figure setting screen 1585
12.3.36.7 Rectangular parallelepiped setting screen.....1601
12.3.36.8 Cylinder setting screen.....1602
12.3.36.9 Plane setting screen.....1603
12.3.36.10 Shape number list screen.....1605
12.3.36.11 Interference check valid figure selection screen 1607
12.3.36.12 Moving axis setting menu screen and moving axis setting screen.....1609
12.3.36.13 Setting screens 1612
12.3.36.14 Name setting screen 1612
12.3.36.15 Display setting screen 1614
12.3.36.16 Drawing coordinate system setting screen.....1615
12.3.36.17 Input and output setting data for 3D interference check 1617
12.3.37 Setting and Displaying Data When the Tool Offset for Milling and Turning Function Is Enabled (15/19-inch Display Unit) 1624
12.4 SCREENS DISPLAYED BY FUNCTION KEY SYSTEM 1627
12.4.1 Displaying and Setting Parameters.....1627
12.4.2 Servo Parameters.....1630
12.4.3 Servo Tuning.....1631
12.4.4 Spindle Setting 1632
12.4.5 Spindle Tuning 1633
12.4.6 Spindle Monitor.....1634
12.4.7 Color Setting Screen.....1635
12.4.8 Machining Parameter Tuning.....1636
12.4.8.1 Machining parameter tuning (AI contour) 1636
12.4.8.2 Machining parameter tuning (nano smoothing).....1642
12.4.9 Displaying Memory Data 1644
12.4.10 Parameter Tuning Screen 1646
12.4.10.1 Displaying the menu screen and selecting a menu item.....1646
12.4.10.2 Parameter tuning screen (system setting).....1649
12.4.10.3 Parameter tuning screen (axis setting) 1650
12.4.10.4 Displaying and setting the FSSB servo amplifier setting screen.....1651
12.4.10.5 Displaying and setting the FSSB spindle amplifier setting screen.....1651
12.4.10.6 Displaying and setting the FSSB axis setting screen 1652
12.4.10.7 Displaying and setting the servo setting screen 1652
12.4.10.8 Parameter tuning screen (spindle setting) 1653
12.4.10.9 Parameter tuning screen (miscellaneous settings).....1653
12.4.10.10 Displaying and setting the servo tuning screen.....1654
12.4.10.11 Displaying and setting the spindle tuning screen.....1654
12.4.10.12 Displaying and setting the machining parameter tuning screen.....1655
12.4.11 Periodic Maintenance Screen 1658
12.4.12 System Configuration Screen.....1667
12.4.13 Power Consumption Monitoring Screen 1669
12.4.14 Displaying and Setting Parameters (15/19-inch Display Unit) 1672

12.4.15 Servo Parameters (15/19-inch Display Unit) 1674
12.4.16 Servo Tuning (15/19-inch Display Unit)....1675
12.4.17 Spindle Setting (15/19-inch Display Unit)....1676
12.4.18 Spindle Tuning (15/19-inch Display Unit)....1677
12.4.19 Spindle Monitor (15/19-inch Display Unit)....1678
12.4.20 Color Setting Screen (15/19-inch Display Unit)....1679
12.4.21 Machining Parameter Tuning (15/19-inch Display Unit)....1680
12.4.21.1 Machining parameter tuning (AI contour)....1680
12.4.21.2 Machining parameter tuning (nano smoothing)....1685
12.4.22 Displaying Memory Data (15/19-inch Display Unit)....1687
12.4.23 Parameter Tuning Screen (15/19-inch Display Unit)....1690
12.4.23.1 Displaying the menu screen and selecting a menu item (15/19-inch display unit)....1690
12.4.23.2 Parameter tuning screen (system setting) (15/19-inch display unit)....1693
12.4.23.3 Parameter tuning screen (axis setting) (15/19-inch display unit)....1694
12.4.23.4 Displaying and setting the FSSB servo amplifier setting screen (15/19-inch display unit)....1695
12.4.23.5 Displaying and setting the FSSB spindle amplifier setting screen (15/19-inch display unit)....1695
12.4.23.6 Displaying and setting the FSSB axis setting screen (15/19-inch display unit)....1696
12.4.23.7 Displaying and setting the servo setting screen (15/19-inch display unit)....1696
12.4.23.8 Parameter tuning screen (spindle setting) (15/19-inch display unit)....1697
12.4.23.9 Parameter tuning screen (miscellaneous settings) (15/19-inch display unit)1697
12.4.23.10 Displaying and setting the servo tuning screen (15/19-inch display unit)....1698
12.4.23.11 Displaying and setting the spindle tuning screen (15/19-inch display unit)1698
12.4.23.12 Displaying and setting the machining parameter tuning screen (15/19-inch display unit)....1699
12.4.24 Periodic Maintenance Screen (15/19-inch Display Unit)....1703
12.4.25 System Configuration Screen (15/19-inch Display Unit)....1710
12.4.26 Power Consumption Monitoring Screen (15/19-inch Display Unit)....1713
12.5 SCREENS DISPLAYED BY FUNCTION KEY  1715
12.5.1 External Operator Message History....1715
12.6 SWITCHING BETWEEN MULTI-PATH DISPLAY AND SINGLE-PATH DISPLAY FUNCTION....1718
12.7 FIVE AXES DISPLAY IN ONE SCREEN FOR THE 8.4-INCH DISPLAY UNIT....1722
12.8 PATH NAME EXPANSION DISPLAY FUNCTION....1725
12.9 SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION....1727
12.10 LOAD METER SCREEN....1729
12.10.1 Single-path Display....1729
12.10.2 Two-path Display and Three-path Display....1730
12.11 DISPLAYING THE PROGRAM NUMBER/NAME, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION....1734
12.11.1 Displaying the Program Number, Program Name, and Sequence Number....1734
12.11.2 Displaying the Status and Warning for Data Setting or Input/Output Operation....1735
12.11.3 Displaying the Program Number, Program Name, and Sequence Number (15/19-inch Display Unit)....1738
12.11.4 Displaying the Program Name....1739

12.11.5 Displaying the Status and Warning for Data Setting or Input/Output Operation (15/19-inch Display Unit) 1740

13 GRAPHIC FUNCTION....1743
13.1 GRAPHIC DISPLAY 1743
13.2 DYNAMIC GRAPHIC DISPLAY 1754
13.2.1 Path Drawing....1754
13.2.1.1 GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen....1754
13.2.1.2 PATH GRAPHIC screen 1759
13.2.1.3 PATH GRAPHIC (TOOL POSITION) screen....1765
13.2.2 Animation....1767
13.2.2.1 GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen....1767
13.2.2.2 ANIMATION GRAPHIC screen....1770
13.2.3 Programmable Data Input (G10) for Blank Figure Drawing Parameters....1776
13.2.4 Warning Messages 1777
13.2.5 Note 1778
13.2.6 Restrictions....1778

14 VIRTUAL MDI KEY FUNCTION....1786
14.1 VIRTUAL MDI KEY....1786
14.1.1 Limitations....1789

15 TEMPLATE PROGRAM FUNCTION 1790
15.1 Template Program Function 1790
15.1.1 Details....1790
15.1.2 Operation....1794
15.1.3 Protection Function 1798
15.1.4 Limitations....1800

16 MULTI-PATH PROGRAM MANAGEMENT FUNCTION....1801
16.1 MULTI-PATH PROGRAM MANAGEMENT FUNCTION....1801
16.1.1 Details....1801
16.1.1.1 Multi-path program folder 1802
16.1.1.2 Making of machining programs....1802
16.1.1.3 Edit of multi-path program 1804
16.1.1.4 Selection of machining program....1804
16.1.1.5 Input/output of Multi-path program....1805
16.1.1.6 Name change of the multi-path program folder....1808
16.1.2 Operation of inputting and editing for each program....1808
16.1.3 Limitations....1810

IV. MAINTENANCE

1 ROUTINE MAINTENANCE 1813
1.1 ACTION TO BE TAKEN WHEN A PROBLEM OCCURRED 1814
1.2 BACKING UP VARIOUS DATA ITEMS 1815
1.3 METHOD OF REPLACING BATTERY 1817
1.3.1 Replacing Battery for Control Unit....1818
1.3.2 Battery in the PANEL *i* (3 VDC) 1821
1.3.3 Replacing Battery for Absolute Pulsecoders....1822
1.3.3.1 Overview....1822
1.3.3.2 Replacing batteries....1822
1.3.3.3 Replacing the batteries in a separate battery case 1823
1.3.3.4 Replacing the battery built into the servo amplifier....1823

APPENDIX

A PARAMETERS 1827
A.1 DESCRIPTION OF PARAMETERS 1827
A.2 DATA TYPE 2146
A.3 STANDARD PARAMETER SETTING TABLES 2147

B PROGRAM CODE LIST 2149

C LIST OF FUNCTIONS AND PROGRAM FORMAT 2152

D RANGE OF COMMAND VALUE 2164

E NOMOGRAPHS 2167
E.1 INCORRECT THREADED LENGTH 2167
E.2 SIMPLE CALCULATION OF INCORRECT THREAD LENGTH 2168
E.3 TOOL PATH AT CORNER 2170
E.4 RADIUS DIRECTION ERROR AT CIRCLE CUTTING 2173

F SETTINGS AT POWER-ON, IN THE CLEAR STATE, OR IN THE RESET STATE 2174

G CHARACTER-TO-CODES CORRESPONDENCE TABLE 2177
G.1 CHARACTER-TO-CODES CORRESPONDENCE TABLE 2177
G.2 FANUC DOUBLE-BYTE CHARACTER CODE TABLE 2178

H ALARM LIST 2184

I PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING 2266
I.1 PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING ... 2266
I.1.1 Usage Notes 2266
I.1.2 List of Functions of PC Tool 2266
I.1.3 Explanation of Operations 2267
I.2 NAMING RULES 2275
I.2.1 Naming Rules of Program File 2275
I.2.2 Naming Rules of Folder 2275
I.3 RULES OF CHARACTERS IN PROGRAM FILE 2276
I.3.1 Usable Characters in Program File 2276
I.4 ERROR MESSAGE AND NOTE 2277
I.4.1 List of Error Message 2277
I.4.2 Note 2278

J ISO/ASCII CODE CONVERSION TOOL 2279

I. GENERAL

1 GENERAL

This manual consists of the following parts:

About this manual

I. GENERAL
Describes chapter organization, applicable models, related manuals, and notes for reading this manual.

II. PROGRAMMING
Describes each function: Format used to program functions in the NC language, explanations, and limitations.

III. OPERATION
Describes the manual operation and automatic operation of a machine, procedures for inputting and outputting data, and procedures for editing a program.

IV. MAINTENANCE
Describes procedures for daily maintenance and replacing batteries.

APPENDIX
Lists parameters, valid data ranges, and alarms.

NOTE
1 This manual describes the functions common to the lathe system and machining center system. For the functions specific to the lathe system or machining center system, refer to the OPERATOR'S MANUAL (Lathe System) (B-64484EN-1) or the OPERATOR'S MANUAL (Machining Center System) (B-64484EN-2).
2 Some functions described in this manual may not be applied to some products. For detail, refer to the Descriptions manual (B-64482EN).
3 This manual does not detail the parameters not mentioned in the text. For details of those parameters, refer to the Parameter Manual (B-64490EN). Parameters are used to set functions and operating conditions of a CNC machine tool, and frequently-used values in advance. Usually, the machine tool builder factory-sets parameters so that the user can use the machine tool easily.
4 This manual describes not only basic functions but also optional functions. Look up the options incorporated into your system in the manual written by the machine tool builder.

Applicable models

This manual describes the models indicated in the table below.
In the text, the abbreviations indicated below may be used.

Model name	Abbreviation	
FANUC Series 30*i*-B	30*i* –B	Series 30*i*
FANUC Series 31*i*-B	31*i* –B	Series 31*i*
FANUC Series 31*i*-B5	31*i* –B5	
FANUC Series 32*i*-B	32*i* –B	Series 32*i*

NOTE
1 For an explanatory purpose, the following descriptions may be used according to the types of path control used:
- T series: For the lathe system
- M series: For the machining center system

NOTE
2 Unless otherwise noted, the model names 31*i*-B, 31*i*-B5, and 32*i*-B are collectively referred to as 30*i*. However, this convention is not necessarily observed when item 3 below is applicable.
3 Some functions described in this manual may not be applied to some products. For details, refer to the Descriptions (B-64482EN).

Special symbols

This manual uses the following symbols:

- **M**

Indicates a description that is valid only for the machine center system set as system control type (in parameter No. 0983).
In a general description of the method of machining, a machining center system operation is identified by a phase such as "for milling machining".

- **T**

Indicates a description that is valid only for the lathe system set as system control type (in parameter No. 0983).
In a general description of the method of machining, a lathe system operation is identified by a phrase such as "for lathe cutting".

-

Indicates the end of a description of a system control type.
When a system control type mark mentioned above is not followed by this mark, the description of the system control type is assumed to continue until the next item or paragraph begins. In this case, the next item or paragraph provides a description common to the control types.

- **IP**

Indicates a combination of axes such as X_ Y_ Z_
In the underlined position following each address, a numeric value such as a coordinate value is placed (used in PROGRAMMING.).

- **;**

Indicates the end of a block. It actually corresponds to the ISO code LF or EIA code CR.

Related manuals of Series 30*i*- MODEL B Series 31*i*- MODEL B Series 32*i*- MODEL B

The following table lists the manuals related to Series 30*i*-B, Series 31*i*-B, Series 32*i*-B. This manual is indicated by an asterisk(*).

Table 1 Related manuals

Manual name	Specification number	
DESCRIPTIONS	B-64482EN	
CONNECTION MANUAL (HARDWARE)	B-64483EN	
CONNECTION MANUAL (FUNCTION)	B-64483EN-1	
OPERATOR'S MANUAL (Common to Lathe System/Machining Center System)	B-64484EN	*
OPERATOR'S MANUAL (For Lathe System)	B-64484EN-1	
OPERATOR'S MANUAL (For Machining Center System)	B-64484EN-2	

Manual name	Specification number	
MAINTENANCE MANUAL	B-64485EN	
PARAMETER MANUAL	B-64490EN	
Programming		
Macro Executor PROGRAMMING MANUAL	B-63943EN-2	
Macro Compiler PROGRAMMING MANUAL	B-66263EN	
C Language Executor PROGRAMMING MANUAL	B-63943EN-3	
PMC		
PMC PROGRAMMING MANUAL	B-64513EN	
Network		
PROFIBUS-DP Board CONNECTION MANUAL	B-63993EN	
Fast Ethernet / Fast Data Server OPERATOR'S MANUAL	B-64014EN	
DeviceNet Board CONNECTION MANUAL	B-64043EN	
FL-net Board CONNECTION MANUAL	B-64163EN	
CC-Link Board CONNECTION MANUAL	B-64463EN	
Operation guidance function		
MANUAL GUIDE *i* (Common to Lathe System/Machining Center System) OPERATOR'S MANUAL	B-63874EN	
MANUAL GUIDE *i* (For Machining Center System) OPERATOR'S MANUAL	B-63874EN-2	
MANUAL GUIDE *i* (Set-up Guidance Functions) OPERATOR'S MANUAL	B-63874EN−1	
Dual Check Safety		
Dual Check Safety CONNECTION MANUAL	B-64483EN-2	

Related manuals of SERVO MOTOR α*i*/β*i* series

The following table lists the manuals related to SERVO MOTOR α*i*/β*i* series

Table 2 Related manuals

Manual name	Specification number
FANUC AC SERVO MOTOR α*i* series DESCRIPTIONS	B-65262EN
FANUC AC SPINDLE MOTOR α*i* series DESCRIPTIONS	B-65272EN
FANUC AC SERVO MOTOR β*i* series DESCRIPTIONS	B-65302EN
FANUC AC SPINDLE MOTOR β*i* series DESCRIPTIONS	B-65312EN
FANUC SERVO AMPLIFIER α*i* series DESCRIPTIONS	B-65282EN
FANUC SERVO AMPLIFIER β*i* series DESCRIPTIONS	B-65322EN
FANUC SERVO MOTOR α*i*s series FANUC SERVO MOTOR α*i* series FANUC AC SPINDLE MOTOR α*i* series FANUC SERVO AMPLIFIER α*i* series MAINTENANCE MANUAL	B-65285EN
FANUC SERVO MOTOR β*i*s series FANUC AC SPINDLE MOTOR β*i* series FANUC SERVO AMPLIFIER β*i* series MAINTENANCE MANUAL	B-65325EN
FANUC AC SERVO MOTOR α*i* series FANUC AC SERVO MOTOR β*i* series FANUC LINEAR MOTOR L*i*S series FANUC SYNCHRONOUS BUILT-IN SERVO MOTOR D*i*S series PARAMETER MANUAL	B-65270EN
FANUC AC SPINDLE MOTOR α*i*/β*i* series, BUILT-IN SPINDLE MOTOR B*i* series PARAMETER MANUAL	B-65280EN

The above servo motors and the corresponding spindles can be connected to the CNC covered in this manual. In the α*i* SV, α*i* SP, α*i* PS, and β*i* SV series, however, they can be connected only to 30 *i*-B-compatible versions. In the β*i* SVSP series, they cannot be connected.

This manual mainly assumes that the FANUC SERVO MOTOR αi series of servo motor is used. For servo motor and spindle information, refer to the manuals for the servo motor and spindle that are actually connected.

1.1 NOTES ON READING THIS MANUAL

⚠ CAUTION

1 The function of an CNC machine tool system depends not only on the CNC, but on the combination of the machine tool, its magnetic cabinet, the servo system, the CNC, the operator's panels, etc. It is too difficult to describe the function, programming, and operation relating to all combinations. This manual generally describes these from the stand-point of the CNC. So, for details on a particular CNC machine tool, refer to the manual issued by the machine tool builder, which should take precedence over this manual.

2 In the header field of each page of this manual, a chapter title is indicated so that the reader can reference necessary information easily.
By finding a desired title first, the reader can reference necessary parts only.

3 This manual describes as many reasonable variations in equipment usage as possible. It cannot address every combination of features, options and commands that should not be attempted.
If a particular combination of operations is not described, it should not be attempted.

1.2 NOTES ON VARIOUS KINDS OF DATA

⚠ CAUTION

Machining programs, parameters, offset data, etc. are stored in the CNC unit internal non-volatile memory. In general, these contents are not lost by the switching ON/OFF of the power. However, it is possible that a state can occur where precious data stored in the non-volatile memory has to be deleted, because of deletions from a maloperation, or by a failure restoration. In order to restore rapidly when this kind of mishap occurs, it is recommended that you create a copy of the various kinds of data beforehand.
The number of times to write machining programs to the non-volatile memory is limited.
You must use "High-speed program management" when registration and the deletion of the machining programs are frequently repeated in such case that the machining programs are automatically downloaded from a personal computer at each machining.
In "High-speed program management", the program is not saved to the non-volatile memory at registration, modification, or deletion of programs.

II. PROGRAMMING

1 GENERAL

Chapter 1, "GENERAL", consists of the following sections:

1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE-INTERPOLATION 9
1.2 FEED-FEED FUNCTION 11
1.3 PART DRAWING AND TOOL MOVEMENT 12
1.4 CUTTING SPEED - SPINDLE FUNCTION 21
1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING - TOOL FUNCTION 22
1.6 COMMAND FOR MACHINE OPERATIONS - AUXILIARY FUNCTION 23
1.7 PROGRAM CONFIGURATION 24
1.8 TOOL MOVEMENT RANGE - STROKE 26

1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE-INTERPOLATION

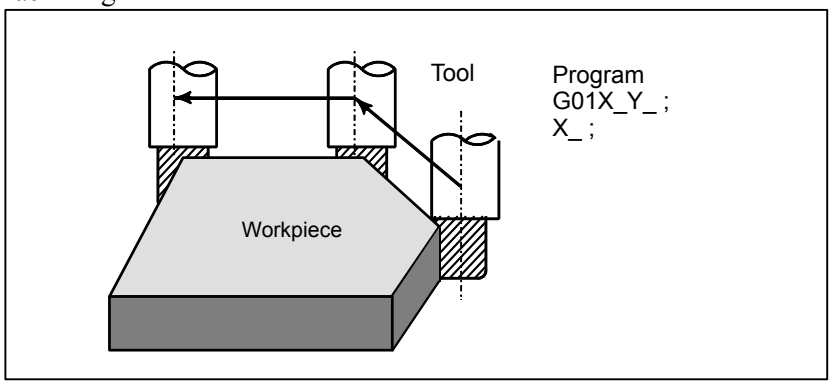
The tool moves along straight lines and arcs constituting the workpiece parts figure (See Chapter, “INTERPOLATION FUNCTIONS”).

Explanation

The function of moving the tool along straight lines and arcs is called the interpolation.

- Tool movement along a straight line

- For milling machining



- For lathe cutting

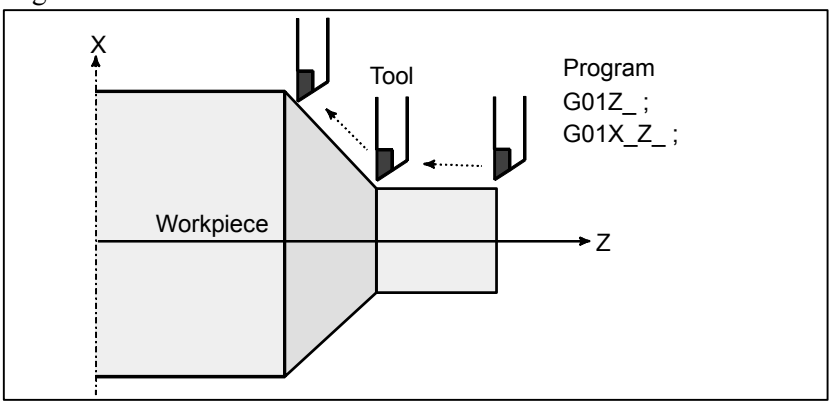
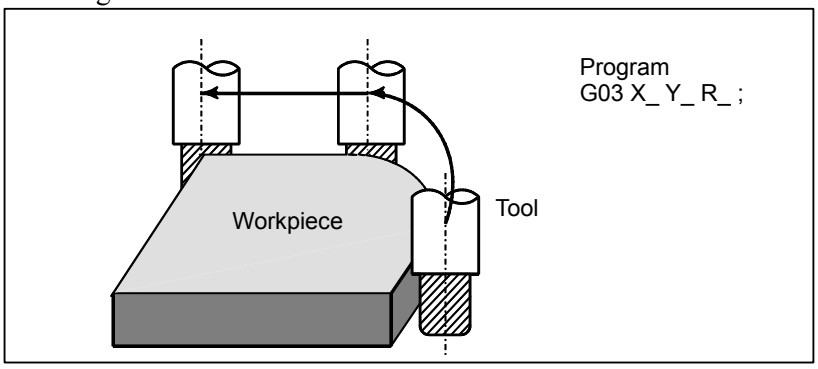


Fig. 1.1 (a) Tool movement along a straight line

- Tool movement along an arc

- For milling machining



- For lathe cutting

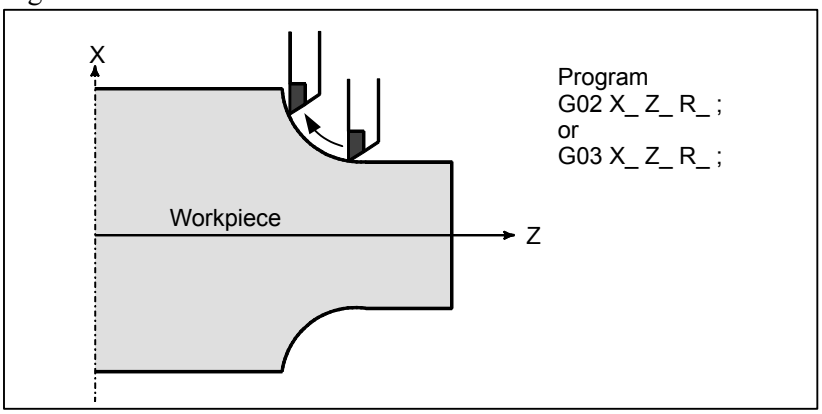


Fig. 1.1 (b) Tool movement along an arc

The term interpolation refers to an operation in which the tool moves along a straight line or arc in the way described above.
Symbols of the programmed commands G01, G02, ... are called the preparatory function and specify the type of interpolation conducted in the control unit.

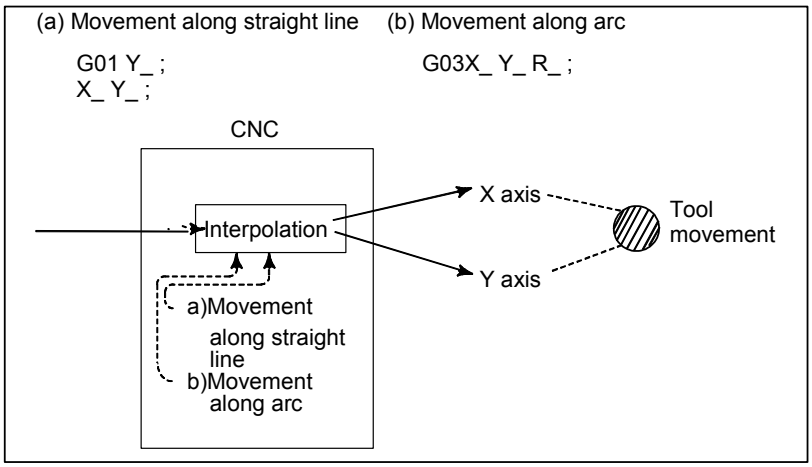


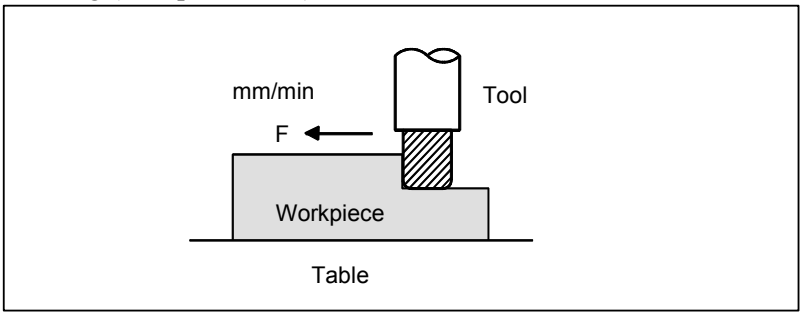
Fig. 1.1 (c) Interpolation function

NOTE
Some machines move tables instead of tools but this manual assumes that tools are moved against workpieces.

1.2 FEED-FEED FUNCTION

Movement of the tool at a specified speed for cutting a workpiece is called the feed.

- For milling machining (feed per minute)



- For lathe cutting (feed per revolution)

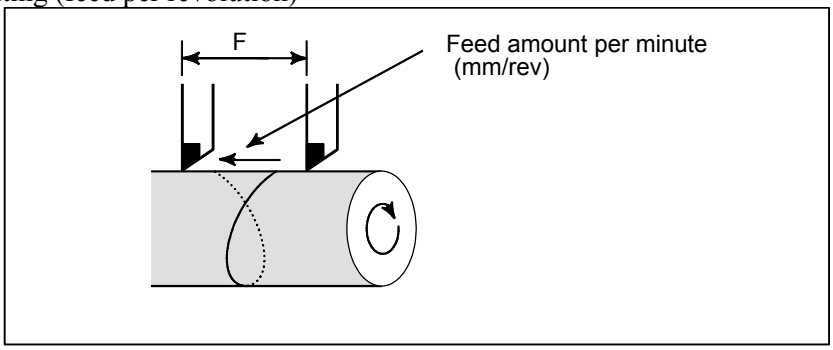


Fig. 1.2 (a) Feed function

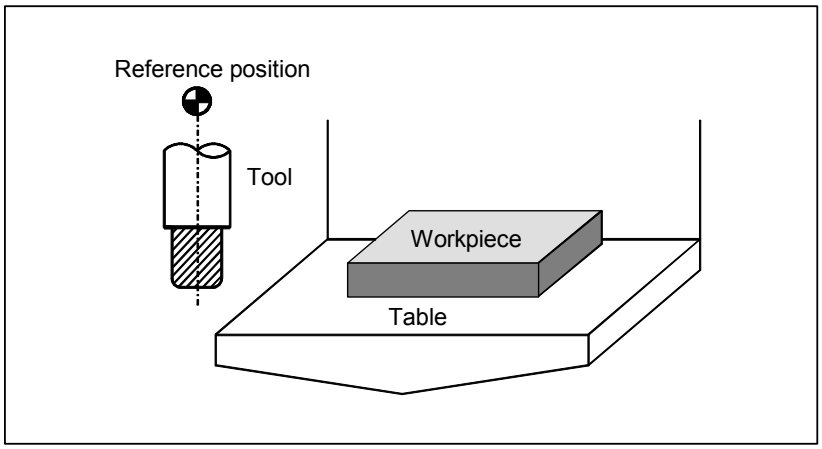
For example, to feed the tool at a rate of 150 mm/min (feed per minute) or 150 mm/rev (feed per revolution), specify the following in the program:
F150.0
The function of deciding the feed rate is called the feed function (See Chapter, “FEED FUNCTIONS”).

1.3 PART DRAWING AND TOOL MOVEMENT

1.3.1 Reference Position (Machine-specific Position)

A CNC machine tool is provided with a fixed position. Normally, tool change and programming of absolute zero point as described later are performed at this position. This position is called the reference position.

- For milling machining



- For lathe cutting

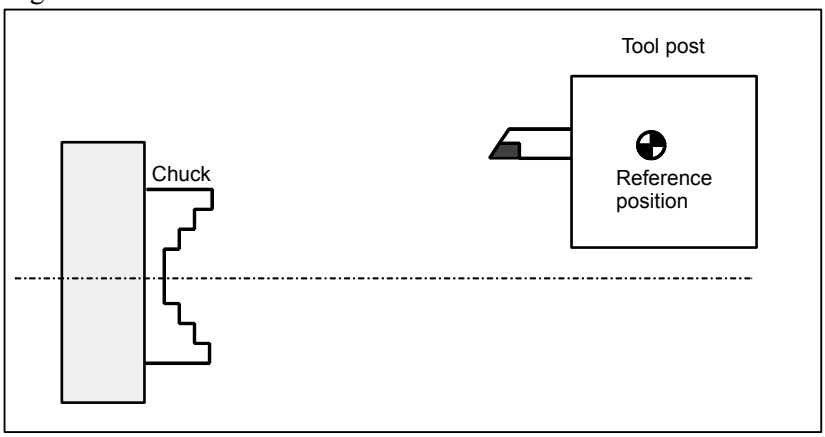


Fig. 1.3.1 (a) Reference position

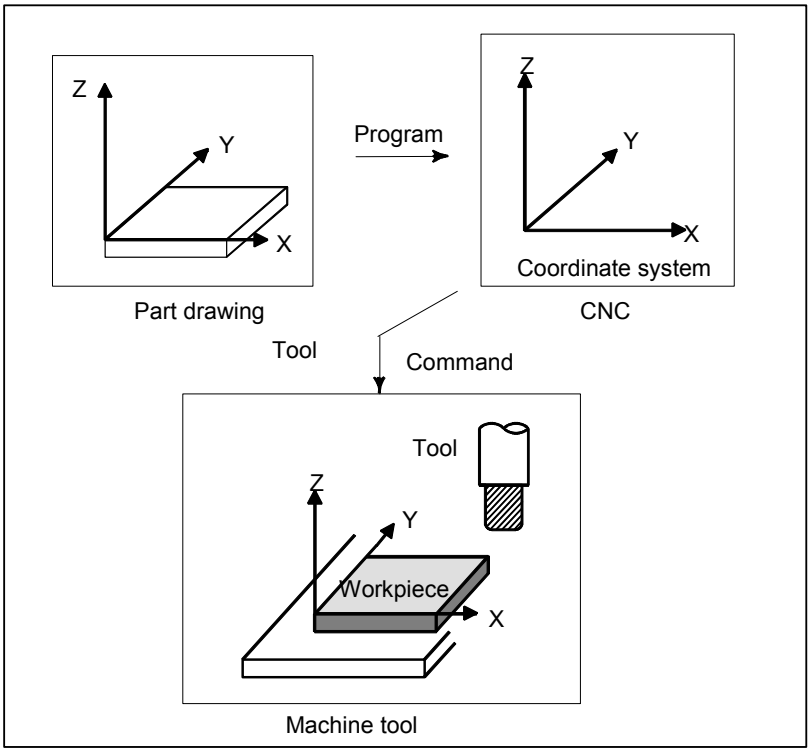
Explanation

The tool can be moved to the reference position in two ways:

1. Manual reference position return (See Section, “MANUAL REFERENCE POSITION RETURN”)
   Reference position return is performed by manual button operation.
2. Automatic reference position return (See Section, “REFERENCE POSITION RETURN”)
   In general, manual reference position return is performed first after the power is turned on. In order to move the tool to the reference position for tool change thereafter, the function of automatic reference position return is used.

1.3.2 Coordinate System on Part Drawing and Coordinate System Specified by CNC - Coordinate System

- For milling machining



- For lathe cutting

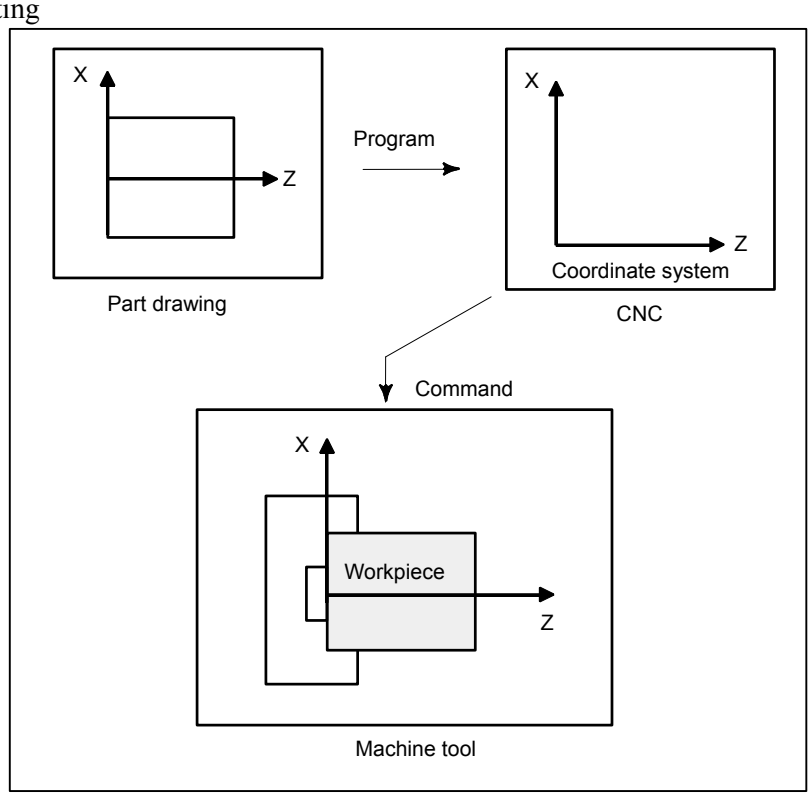


Fig. 1.3.2 (a) Coordinate system

Explanation

- Coordinate system

The following two coordinate systems are specified at different locations:
(See Chapter, "COORDINATE SYSTEM")

1 Coordinate system on part drawing
The coordinate system is written on the part drawing. As the program data, the coordinate values on this coordinate system are used.

2. Coordinate system specified by the CNC
The coordinate system is prepared on the actual machine tool table. This can be achieved by programming the distance from the current position of the tool to the zero point of the coordinate system to be set.

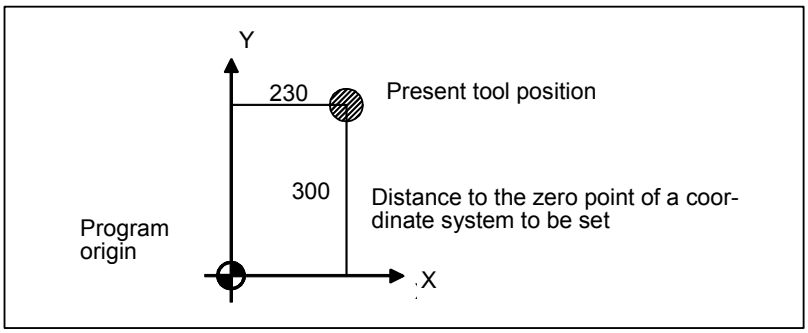
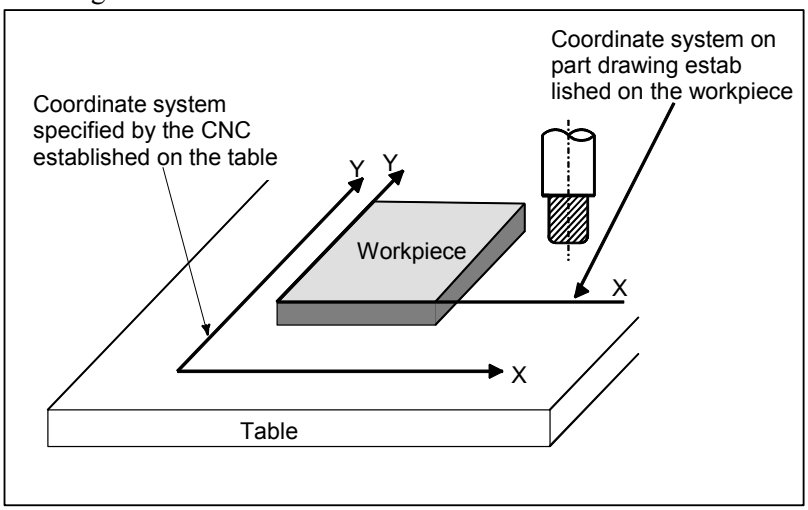


Fig. 1.3.2 (b) Coordinate system specified by the CNC

Concrete programming methods for setting coordinate systems specified by the CNC are explained in Chapter, "COORDINATE SYSTEM"
The positional relation between these two coordinate systems is determined when a workpiece is set on the table.

- For milling machining



- For lathe cutting

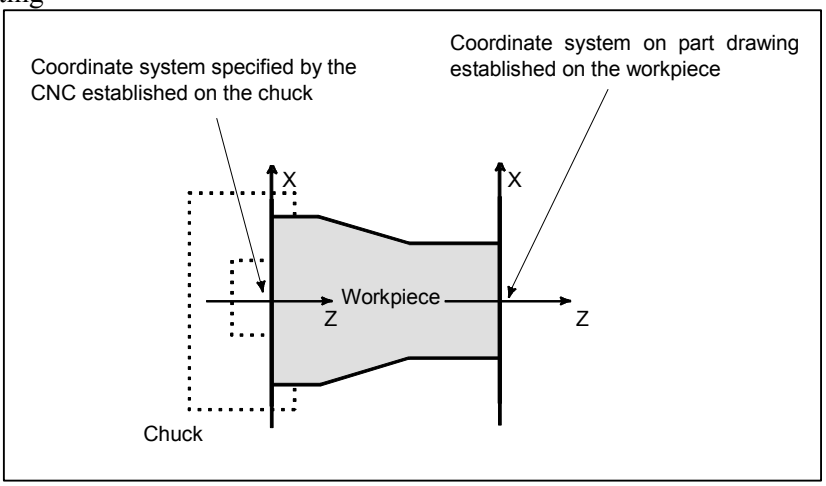


Fig. 1.3.2 (c) Coordinate system specified by CNC and coordinate system on part drawing

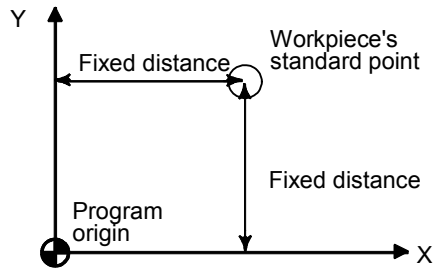
The tool moves on the coordinate system specified by the CNC in accordance with the command program generated with respect to the coordinate system on the part drawing, and cuts a workpiece into a shape on the drawing.
Therefore, in order to correctly cut the workpiece as specified on the drawing, the two coordinate systems must be set at the same position.

- Methods of setting the two coordinate systems in the same position

M

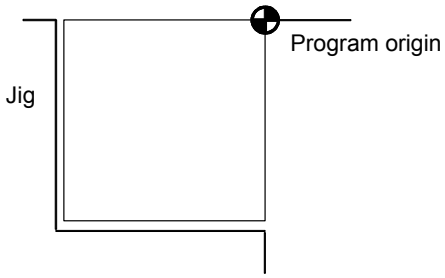
To set the two coordinate systems at the same position, simple methods shall be used according to workpiece shape, the number of machinings.

1. Using a standard plane and point of the workpiece.



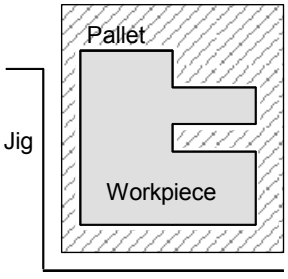
Bring the tool center to the workpiece standard point.
And set the coordinate system specified by CNC at this position.

2. Mounting a workpiece directly against the jig



Meet the tool center to the reference position. And set the coordinate specified by CNC at this position. (Jig shall be mounted on the point from the reference

3. Mounting a workpiece on a pallet, then mounting the workpiece and pallet on the jig

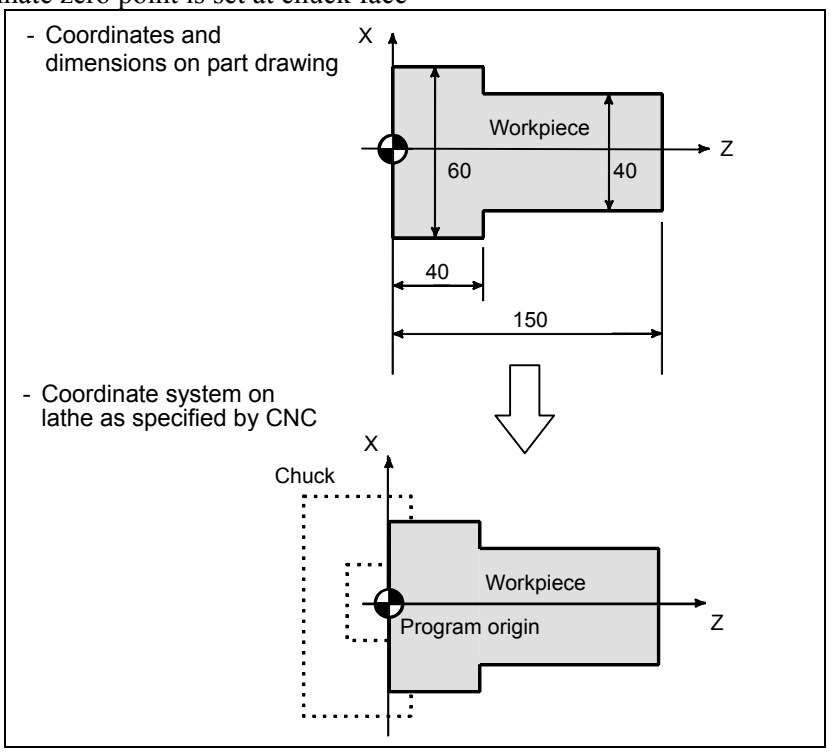


(Jig and coordinate system shall be specified by the same as (2)).

T

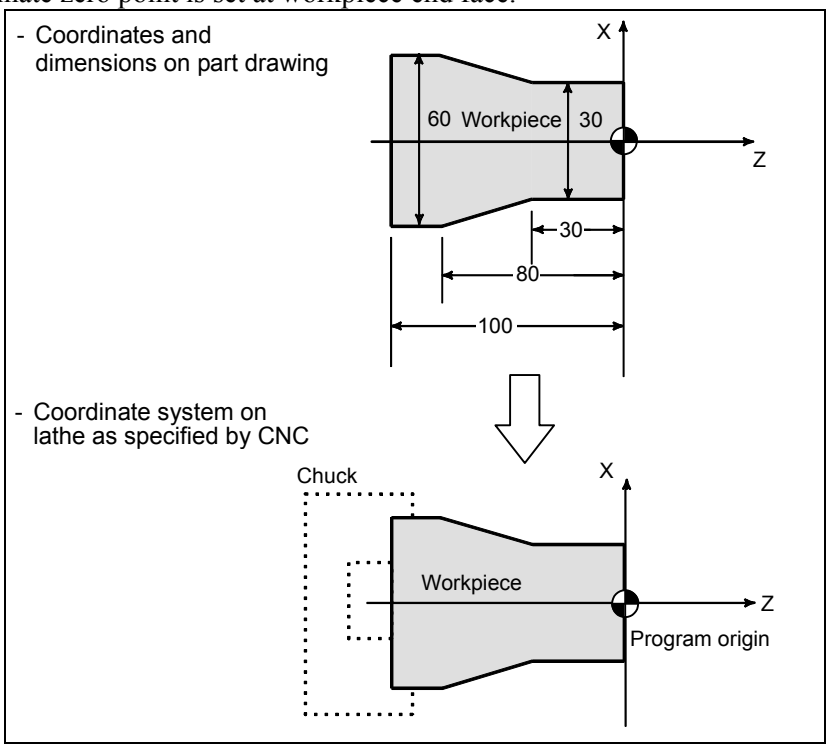
The following method is usually used to define two coordinate systems at the same location.

1 When coordinate zero point is set at chuck face



When the coordinate system on the part drawing and the coordinate system specified by the CNC are set at the same position, the program origin can be set on the chuck face.

2. When coordinate zero point is set at workpiece end face.



When the coordinate system on the part drawing and the coordinate system specified by the CNC are set at the same position, the program origin can be set on the end face of the workpiece.

1.3.3 How to Indicate Command Dimensions for Moving the Tool (Absolute and Incremental Programming)

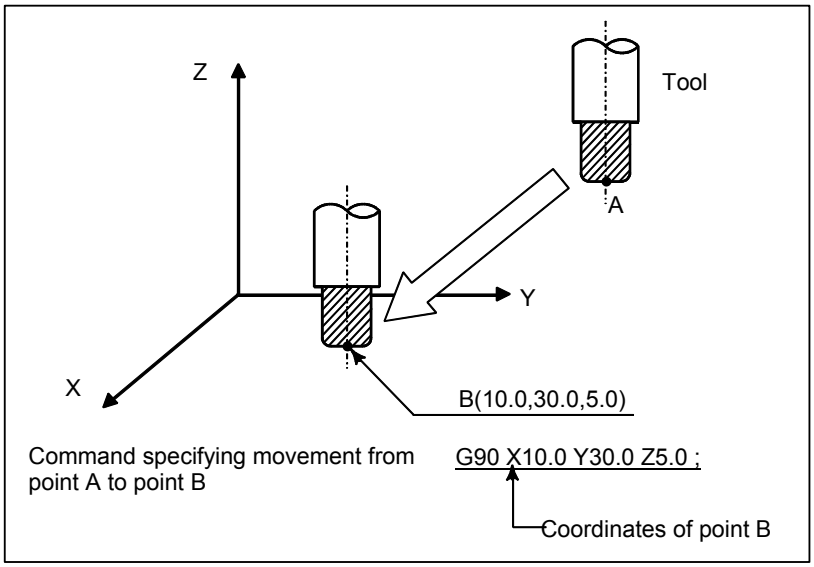
Explanation

Command for moving the tool can be indicated by absolute command or incremental command (See Section, “ABSOLUTE AND INCREMENTAL PROGRAMMING”).

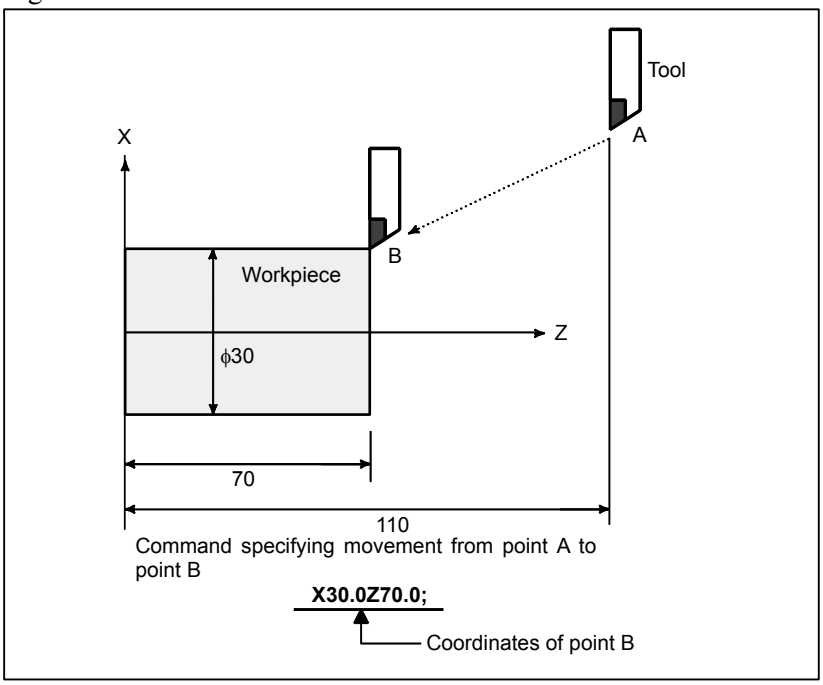
- Absolute command

The tool moves to a point at "the distance from zero point of the coordinate system" that is to the position of the coordinate values.

- For milling machining



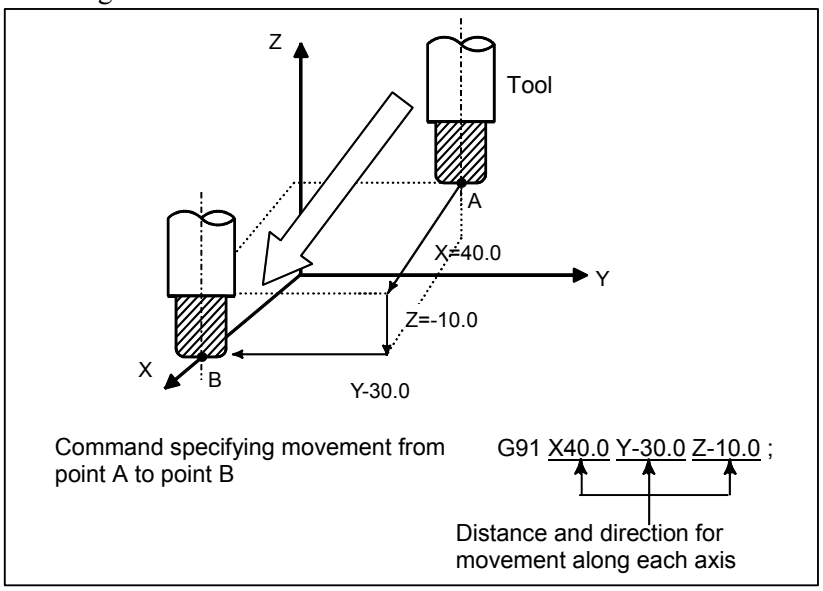
- For lathe cutting



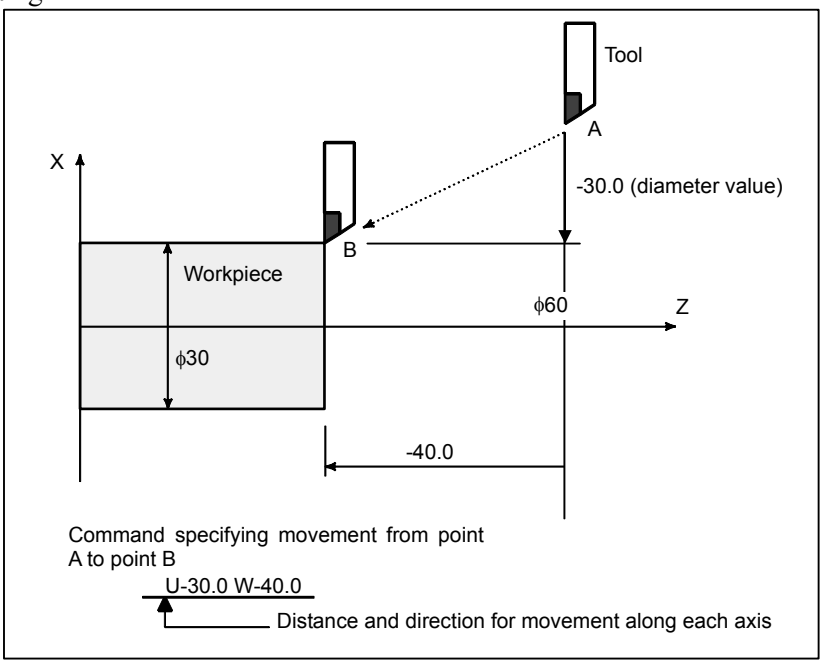
- Incremental command

Specify the distance from the previous tool position to the next tool position.

- For milling machining



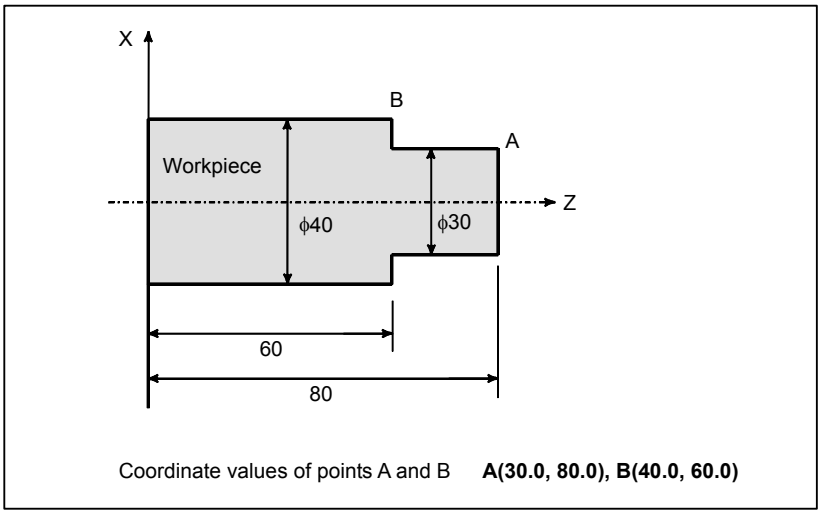
- For lathe cutting



- Diameter programming / radius programming

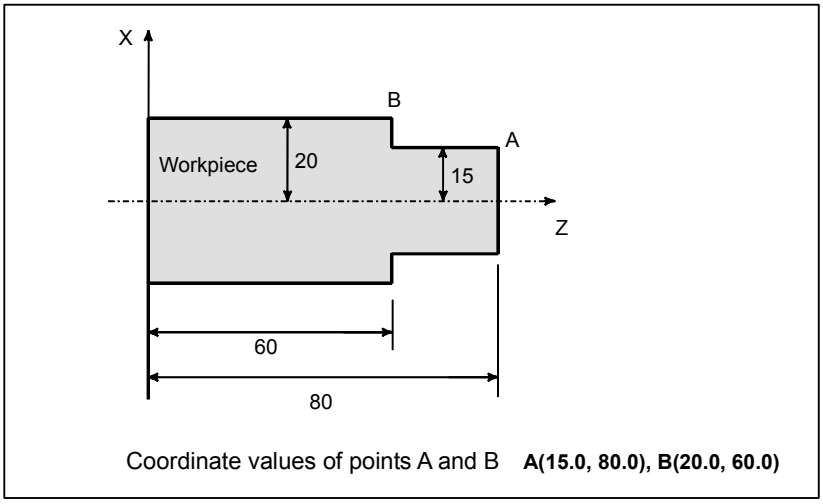
Dimensions of the X-axis can be set in diameter or in radius. Which programming is used is determined according to the setting of bit 3 (DIA) of parameter No. 1006.

1. Diameter programming
   In diameter programming, specify the diameter value indicated on the drawing as the value of the X-axis.



Coordinate values of points A and B **A(30.0, 80.0), B(40.0, 60.0)**

2. Radius programming
   In radius programming, specify the distance from the center of the workpiece, i.e. the radius value as the value of the X-axis.



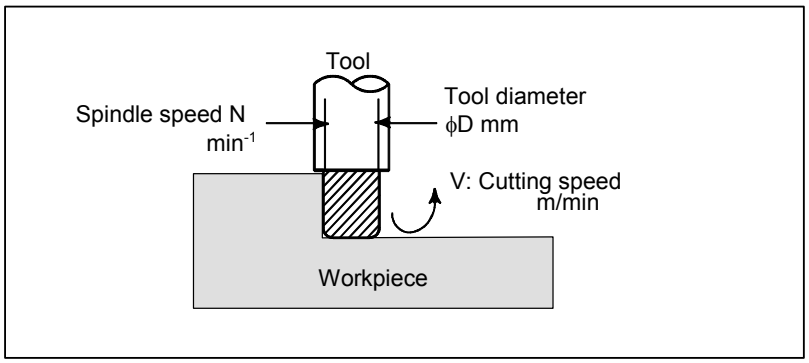
Coordinate values of points A and B **A(15.0, 80.0), B(20.0, 60.0)**

1.4 CUTTING SPEED - SPINDLE FUNCTION

Explanation

The speed of the tool with respect to the workpiece when the workpiece is cut is called the cutting speed. As for the CNC, the cutting speed can be specified by the spindle speed in min^{-1} unit.

- For milling machining

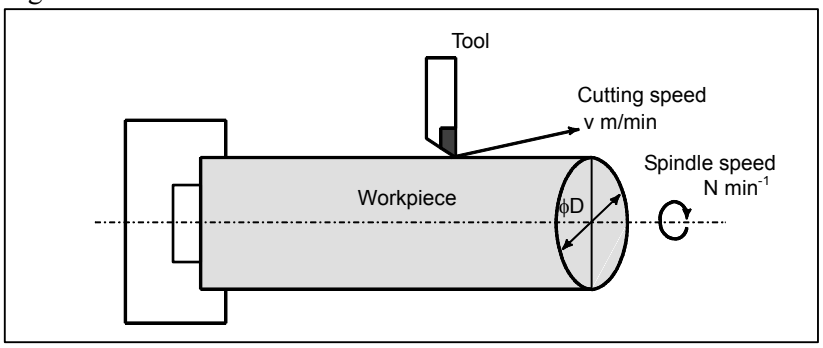


<When a workpiece should be machined with a tool 100 mm in diameter at a cutting speed of 80 m/min.>
The spindle speed is approximately 250 min^{-1}, which is obtained from N=1000v/πD. Hence the following command is required:

S250;

Commands related to the spindle speed are called the spindle speed function (See Chapter, "SPINDLE SPEED FUNCTION (S FUNCTION)").

- For lathe cutting



<When a workpiece 200 mm in diameter should be machined at a cutting speed of 300 m/min.>
The spindle speed is approximately 478 min^{-1}, which is obtained from N=1000v/πD. Hence the following command is required:

S478;

Commands related to the spindle speed are called the spindle speed function (See Chapter, "SPINDLE SPEED FUNCTION (S FUNCTION)").
The cutting speed v (m/min) can also be specified directly by the speed value. Even when the workpiece diameter is changed, the CNC changes the spindle speed so that the cutting speed remains constant.
This function is called the constant surface speed control function (See Section, "CONSTANT SURFACE SPEED CONTROL (G96, G97)").

1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING - TOOL FUNCTION

For each of various types of machining (such as drilling, tapping, boring, and milling for milling machining, or rough machining, semifinish machining, finish machining, threading, and grooving for lathe cutting), a necessary tool is to be selected. When a number is assigned to each tool and the number is specified in the program, the corresponding tool is selected.

Examples

M

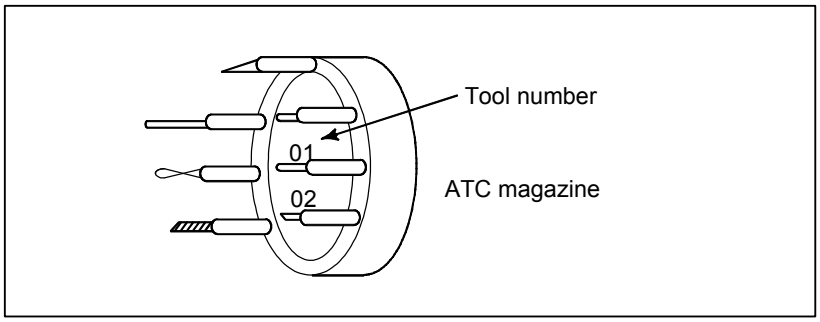


Fig. 1.5 (a) Tool used for various machining

<When No.01 is assigned to a drilling tool>
When the tool is stored at location 01 in the ATC magazine, the tool can be selected by specifying T01. This is called the tool function (See Chapter, “TOOL FUNCTION (T FUNCTION)”).

T

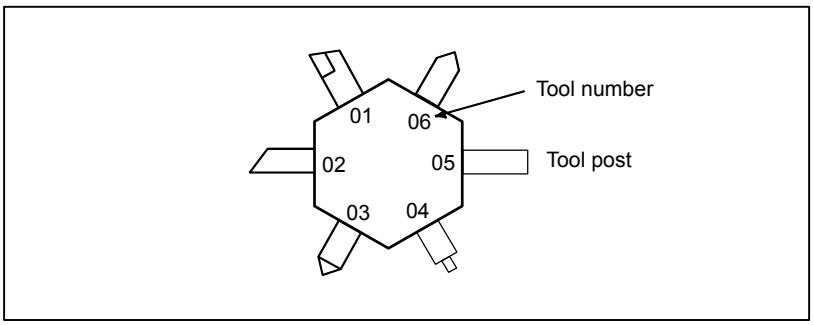


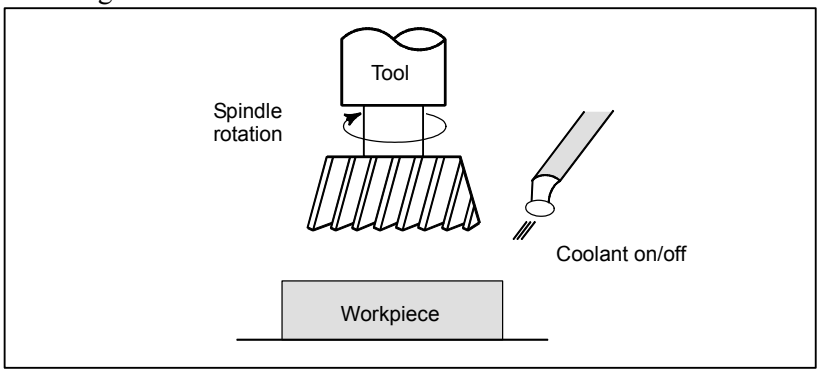
Fig. 1.5 (b) Tool used for various machining

<When No.01 is assigned to a roughing tool>
When the tool is stored at location 01 of the tool post, the tool can be selected by specifying T0101. This is called the tool function (See Chapter, “TOOL FUNCTION (T FUNCTION)”).

1.6 COMMAND FOR MACHINE OPERATIONS - AUXILIARY FUNCTION

When a workpiece is actually machined with a tool, the spindle is rotated, coolant is supplied, and the chuck is opened/closed. So, control needs to be exercised on the spindle motor of the machine, coolant valve on/off operation, and chuck open/close operation.

- For milling machining



- For lathe cutting

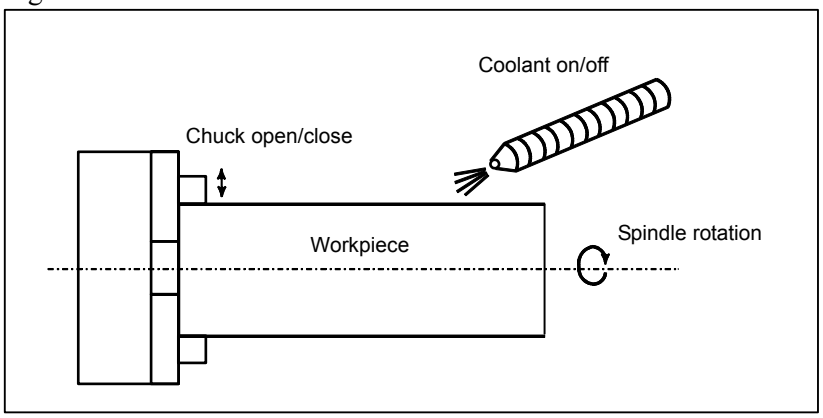


Fig. 1.6 (a) Auxiliary function

The function of specifying the on-off operations of the components of the machine is called the auxiliary function. In general, the function is specified by an M code (See Chapter, “AUXILIARY FUNCTION”). For example, when M03 is specified, the spindle is rotated clockwise at the specified spindle speed.

1.7 PROGRAM CONFIGURATION

A group of commands given to the CNC for operating the machine is called the program. By specifying the commands, the tool is moved along a straight line or an arc, or the spindle motor is turned on and off. In the program, specify the commands in the sequence of actual tool movements.

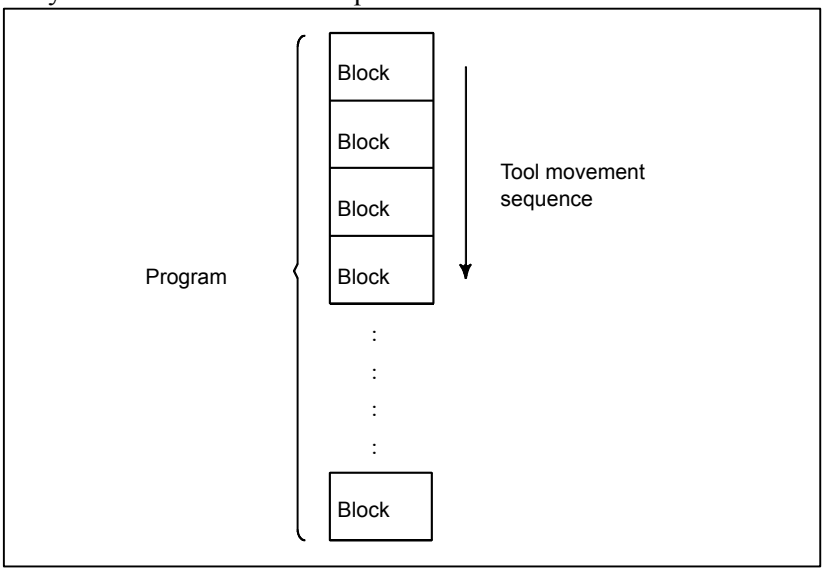


Fig. 1.7 (a) Program configuration

A group of commands at each step of the sequence is called the block. The program consists of a group of blocks for a series of machining. The number for discriminating each block is called the sequence number, and the number for discriminating each program is called the program number (See Chapter, "PROGRAM CONFIGURATION").

Explanation

The block and the program have the following configurations.

- Block

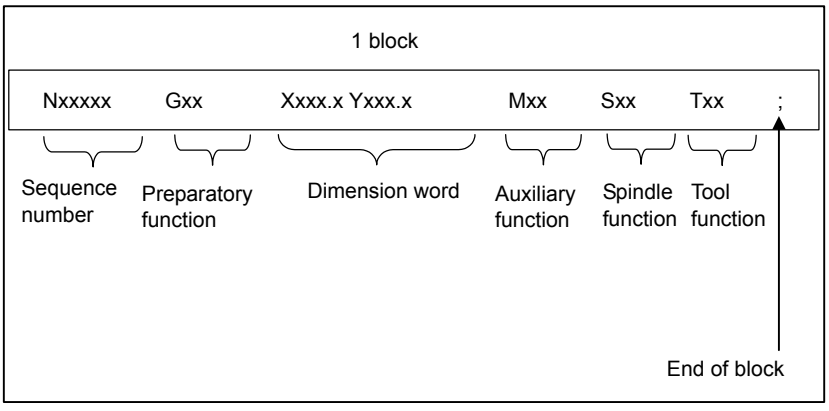


Fig. 1.7 (b) Block configuration

A block starts with a sequence number to identify the block and ends with an end-of-block code.
This manual indicates the end-of-block code by; (LF in the ISO code and CR in the EIA code).
The contents of the dimension word depend on the preparatory function. In this manual, the portion of the dimension word may be represent as IP_.

- Program

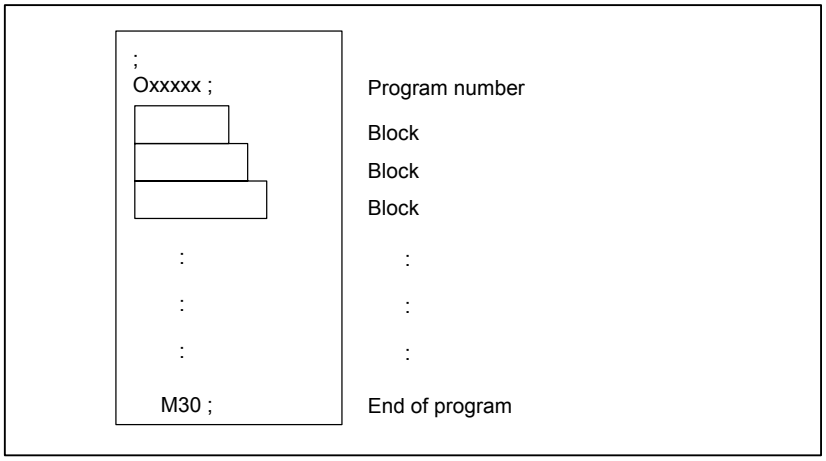


Fig. 1.7 (c) Program configuration

Normally, a program number is specified after the end-of-block (;) code at the beginning of the program, and a program end code (M02 or M30) is specified at the end of the program.

- Main program and subprogram

When machining of the same pattern appears at many portions of a program, a program for the pattern is created. This is called the subprogram. On the other hand, the original program is called the main program. When a subprogram execution command appears during execution of the main program, commands of the subprogram are executed. When execution of the subprogram is finished, the sequence returns to the main program.

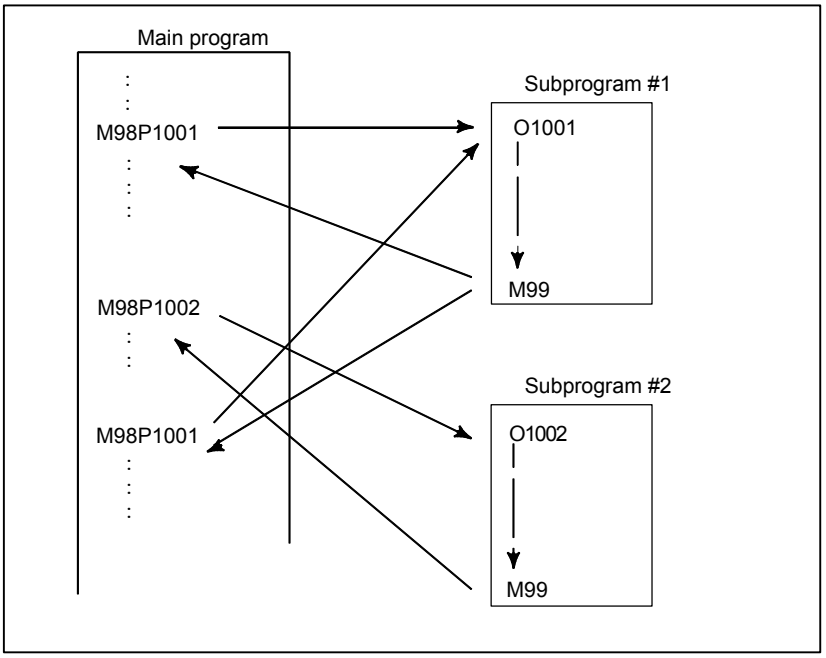
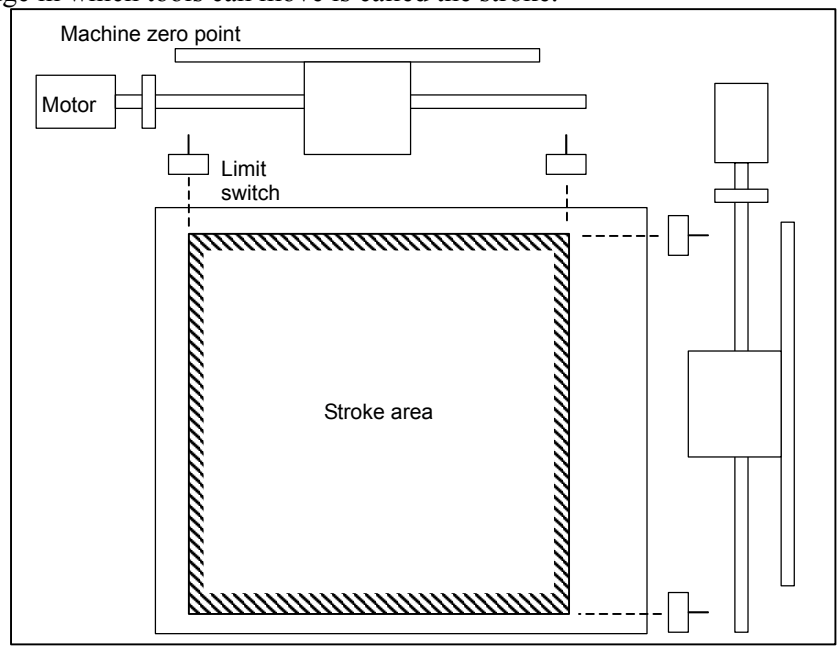


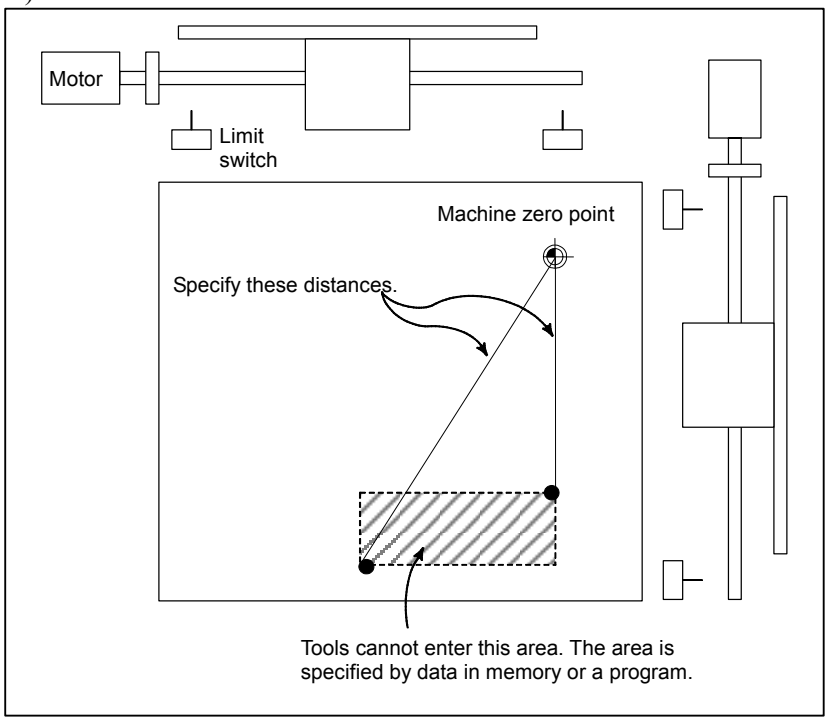
Fig. 1.7 (d) Subprogram execution

1.8 TOOL MOVEMENT RANGE - STROKE

Limit switches are installed at the ends of each axis on the machine to prevent tools from moving beyond the ends. The range in which tools can move is called the stroke.



Besides strokes defined with limit switches, the operator can define an area which the tool cannot enter using a program or data in memory. This function is called stroke check (See Section, “STORED STROKE CHECK”).



2 CONTROLLED AXES

Chapter 2, "CONTROLLED AXES", consists of the following sections:

2.1 NUMBER OF CONTROLLED AXES....27
2.2 NAMES OF AXES....27
2.3 INCREMENT SYSTEM....28
2.4 MAXIMUM STROKE....29

2.1 NUMBER OF CONTROLLED AXES

Explanation

The number of controlled axes used with this NC system depends on the model and system control type as indicated below.

		Series 30*i*-B	Series 31*i*-B5	Series 31*i*-B	Series 32*i*-B
Number of basic controlled axes	Lathe system	2 axes	2 axes	2 axes	2 axes
	Machining center system	3 axes	3 axes	3 axes	3 axes
Controlled axes expansion (each axis) (including Cs axes and PMC axes)		Max. 24 axes	Max. 12 axes	Max. 12 axes	Max. 5 axes
Controlled axes expansion (total) (including Cs axes and PMC axes)		Max. 32 axes	Max. 20 axes	Max. 20 axes	Max. 9 axes
Basic simultaneously controlled axes (each path)		2 axes	2 axes	2 axes	2 axes
Simultaneously controlled axes expansion (total / each path)		Max. 24 axes	Max. 5 axes	Max. 4 axes	Max. 4 axes
Axis control by PMC (not available on Cs axis)		Max. 32 axes	Max. 16 axes	Max. 16 axes	Max. 8 axes
Designation of Spindle axes (each path)		Max. 4 axes	Max. 4 axes	Max. 4 axes	Max. 2 axes
Designation of Spindle axes (total)		Max. 8 axes	Max. 6 axes	Max. 6 axes	Max. 2 axes
Cs contouring control (each path)		Max. 4 axes	Max. 4 axes	Max. 4 axes	Max. 2 axes
Cs contouring control (total)		Max. 8 axes	Max. 6 axes	Max. 6 axes	Max. 2 axes

NOTE
1 The maximum number of controlled axes that can be used is limited depending on the option configuration. Refer to the manual provided by the machine tool builder for details.
2 The number of simultaneously controllable axes for manual operation (jog feed, manual reference position return, or manual rapid traverse) is 1 or 3 (1 when bit 0 (JAX) of parameter No. 1002 is set to 0 and 3 when it is set to 1).

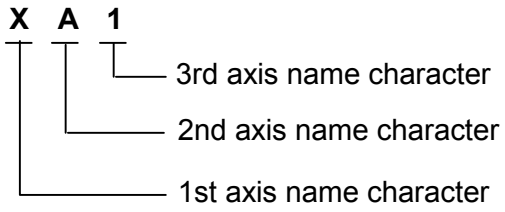
2.2 NAMES OF AXES

Explanation

The move axes of machine tools are assigned names. These names are referred to as addresses or axis names. Axis names are determined according to the machine tool. The naming rules comply with standards such as the ISO standards.

With complex machines, one character would become insufficient for representing axis names. So, up to three characters can be used for axis names. A move axis may be named "X", "X1", or "XA1". The first character of the three characters is called the first axis name character, the second character is called the second axis name character, and third character is called the third axis name character.

Example)



NOTE
1 Axis names are predetermined according to the machine used. Refer to the manual supplied by the machine tool builder.
2 Since many ordinary machines use one character to represent each address, one-character addresses are used in the description in this manual.

2.3 INCREMENT SYSTEM

Explanation

The increment system consists of the least input increment (for input) and least command increment (for output). The least input increment is the least increment for programming the travel distance. The least command increment is the least increment for moving the tool on the machine. Both increments are represented in mm, inches, or deg.
Five types of increment systems are available as indicated in Table 2.3 (a). For each axis, an increment system can be set using a bit from bit 0 to bit 3 (ISA, ISC, ISD, or ISE) of parameter No. 1013.
IS-C, IS-D, and IS-E are optional functions.

Table 2.3 (a) Increment system

Name of increment system	Least input increment		Least command increment	
IS-A	0.01	mm	0.01	mm
	0.001	inch	0.001	inch
	0.01	deg	0.01	deg
IS-B	0.001	mm	0.001	mm
	0.0001	inch	0.0001	inch
	0.001	deg	0.001	deg
IS-C	0.0001	mm	0.0001	mm
	0.00001	inch	0.00001	inch
	0.0001	deg	0.0001	deg
IS-D	0.00001	mm	0.00001	mm
	0.000001	inch	0.000001	inch
	0.00001	deg	0.00001	deg
IS-E	0.000001	mm	0.000001	mm
	0.0000001	inch	0.0000001	inch
	0.000001	deg	0.000001	deg

The least command increment is either metric or inch depending on the machine tool. Set metric or inch to the bit 0 (INM) of parameter No. 0100.
For selection between metric and inch for the least input increment, G code (G20 or G21) or a setting parameter selects it.
Combined use of the inch system and the metric system is not allowed. There are functions that cannot be used between axes with different unit systems (circular interpolation, cutter compensation, etc.). For the increment system, see the machine tool builder's manual.

NOTE
1 The unit (mm or inch) in the table is used for indicating a diameter value for diameter programming (when bit 3 (DIA) of parameter No. 1006 is set to 1) or a radius value for radius programming.
2 Some increment systems are unavailable depending on the model. For details, refer to “Descriptions” (B-64482EN).

2.4 MAXIMUM STROKE

Explanation

The maximum stroke controlled by this CNC is shown in the table below:
Maximum stroke = Least command increment × 999999999 (99999999 for IS-A)
Commands that exceed the maximum stroke are not permitted.

Table 2.4 (a) Maximum strokes

Name of increment system	Least input increment		Maximum stroke	
IS-A	0.01	mm	±999999.99	mm
	0.001	inch	±99999.999	inch
	0.01	deg	±999999.99	deg
IS-B	0.001	mm	±999999.999	mm
	0.0001	inch	±99999.9999	inch
	0.001	deg	±999999.999	deg
IS-C	0.0001	mm	±99999.9999	mm
	0.00001	inch	±9999.99999	inch
	0.0001	deg	±99999.9999	deg
IS-D	0.00001	mm	±9999.99999	mm
	0.000001	inch	±999.999999	inch
	0.00001	deg	±9999.99999	deg
IS-E	0.000001	mm	±999.999999	mm
	0.0000001	inch	±99.9999999	inch
	0.000001	deg	±999.999999	deg

NOTE
1 The actual stroke depends on the machine tool.
2 The unit (mm or inch) in the table is used for indicating a diameter value for diameter programming (when bit 3 (DIA) of parameter No. 1006 is set to 1) or a radius value for radius programming.
3 Some increment systems are unavailable depending on the model. For details, refer to "Descriptions" (B-64482EN).

3 PREPARATORY FUNCTION (G FUNCTION)

A number following address G determines the meaning of the command for the concerned block.
G codes are divided into the following two types.

Type	Meaning
One-shot G code	The G code is effective only in the block in which it is specified.
Modal G code	The G code is effective until another G code of the same group is specified.

(Example)

G01 and G00 are modal G codes in group 01.

```
G01 X_ ;
    Z_ ;  } G01 is effective in this range.
    X_ ;
G00 Z_ ;  } G00 is effective in this range.
    X_ ;
G01 X_ ;
  :
```

T

There are three G code systems in the lathe system: A, B, and C (Table 3.1(a)). Select a G code system using bits 6 (GSB) and 7 (GSC) of parameter No. 3401. To use G code system B or C, the corresponding option is needed. Generally, OPERATOR'S MANUAL describes the use of G code system A, except when the described item can use only G code system B or C. In such cases, the use of G code system B or C is described.

Explanation

1. When the clear state (bit 6 (CLR) of parameter No. 3402) is set at power-up or reset, the modal G codes are placed in the states described below.
   (1) The modal G codes are placed in the states marked with ◪ as indicated in Table.
   (2) G20 and G21 remain unchanged when the clear state is set at power-up or reset.
   (3) Which status G22 or G23 at power on is set by bit 7 (G23) of parameter No. 3402. However, G22 and G23 remain unchanged when the clear state is set at reset.
   (4) The user can select G00 or G01 by setting bit 0 (G01) of parameter No. 3402.
   (5) The user can select G90 or G91 by setting bit 3 (G91) of parameter No. 3402.
   When G code system B or C is used in the lathe system, setting bit 3 (G91) of parameter No. 3402 determines which code, either G90 or G91, is effective.
   (6) In the machining center system, the user can select G17, G18, or G19 by setting bits 1 (G18) and 2 (G19) of parameter No. 3401.
2. G codes other than G10 and G11 are one-shot G codes.
3. When a G code not listed in the G code list is specified, or a G code that has no corresponding option is specified, alarm PS0010 occurs.
4. Multiple G codes can be specified in the same block if each G code belongs to a different group. If multiple G codes that belong to the same group are specified in the same block, only the last G code specified is valid.
5. If a G code belonging to group 01 is specified in a canned cycle for drilling, the canned cycle for drilling is cancelled. This means that the same state set by specifying G80 is set. Note that the G codes in group 01 are not affected by a G code specifying a canned cycle for drilling.
6. G codes are indicated by group.
7. The group of G60 is switched according to the setting of bit 0 (MDL) of parameter No. 5431. (When the MDL bit is set to 0, the 00 group is selected. When the MDL bit is set to 1, the 01 group is selected.)

T

8. When G code system A is used, absolute or incremental programming is specified not by a G code (G90/G91) but by an address word (X/U, Z/W, C/H, Y/V). Only the initial level is provided at the return point of the canned cycle for drilling..

3.1 G CODE LIST IN THE MACHINING CENTER SYSTEM

M

Table 3.1 (a) G code list

G code	Group	Function	
G00	01	Positioning (rapid traverse)	
G01		Linear interpolation (cutting feed)	
G02		Circular interpolation CW or helical interpolation CW	
G03		Circular interpolation CCW or helical interpolation CCW	
G02.1, G03.1		Circular thread cutting B CW/CCW	
G02.2, G03.2		Involute interpolation CW/CCW	
G02.3, G03.3		Exponential interpolation CW/CCW	
G02.4, G03.4		3-dimensional coordinate system conversion CW/CCW	
G04	00	Dwell	
G05		AI contour control (high-precision contour control compatible command), High-speed cycle machining, High-speed binary program operation	
G05.1		AI contour control / Nano smoothing / Smooth interpolation	
G05.4		HRV3, 4 on/off	
G06.2	01	NURBS interpolation	
G07	00	Hypothetical axis interpolation	
G07.1 (G107)		Cylindrical interpolation	
G08		AI contour control (advanced preview control compatible command)	
G09		Exact stop	
G10		Programmable data input	
G10.6		Tool retract and recover	
G10.9		Programmable switching of diameter/radius specification	
G11		Programmable data input mode cancel	
G12.1	21	Polar coordinate interpolation mode	
G13.1		Polar coordinate interpolation cancel mode	
G12.4	00	Groove cutting by continuous circle motion (CW)	
G13.4		Groove cutting by continuous circle motion (CCW)	
G15	17	Polar coordinates command cancel	
G16		Polar coordinates command	
G17	02	XpYp plane selection	Xp: X axis or its parallel axis Yp: Y axis or its parallel axis Zp: Z axis or its parallel axis
G17.1		Plane conversion function	
G18		ZpXp plane selection	
G19		YpZp plane selection	
G20 (G70)	06	Input in inch	
G21 (G71)		Input in mm	
G22	04	Stored stroke check function on	
G23		Stored stroke check function off	
G25	19	Spindle speed fluctuation detection off	
G26		Spindle speed fluctuation detection on	
G27	00	Reference position return check	
G28		Automatic return to reference position	
G28.2		In-position check disable reference position return	
G29		Movement from reference position	
G30		2nd, 3rd and 4th reference position return	
G30.1		Floating reference position return	
G30.2		In-position check disable 2nd, 3rd, or 4th reference position return	
G31		Skip function	
G31.8		EGB-axis skip	

Table 3.1 (a) G code list

G code	Group	Function
G33	01	Threading
G34		Variable lead threading
G35		Circular threading CW
G36		Circular threading CCW
G37	00	Automatic tool length measurement
G38		Tool radius/tool nose radius compensation : preserve vector
G39		Tool radius/tool nose radius compensation : corner circular interpolation
G40	07	Tool radius/tool nose radius compensation : cancel 3-dimensional cutter compensation : cancel
G41		Tool radius/tool nose radius compensation : left 3-dimensional cutter compensation : left
G41.2		3-dimensional cutter compensation : left (type 1)
G41.3		3-dimensional cutter compensation : leading edge offset
G41.4		3-dimensional cutter compensation : left (type 1) (FS16i-compatible command)
G41.5		3-dimensional cutter compensation : left (type 1) (FS16i-compatible command)
G41.6		3-dimensional cutter compensation : left (type 2)
G42		Tool radius/tool nose radius compensation : right 3-dimensional cutter compensation : right
G42.2		3-dimensional cutter compensation : right (type 1)
G42.4		3-dimensional cutter compensation : right (type 1) (FS16i-compatible command)
G42.5		3-dimensional cutter compensation : right (type 1) (FS16i-compatible command)
G42.6		3-dimensional cutter compensation : right (type 2)
G40.1	18	Normal direction control cancel mode
G41.1		Normal direction control on : left
G42.1		Normal direction control on : right
G43	08	Tool length compensation +
G44		Tool length compensation -
G43.1		Tool length compensation in tool axis direction
G43.3		Nutating rotary head tool length compensation
G43.4		Tool center point control (type 1)
G43.5		Tool center point control (type 2)
G43.7		Tool offset
G44.1		Tool offse conversion
G45	00	Tool offset : increase
G46		Tool offset : decrease
G47		Tool offset : double increase
G48		Tool offset : double decrease
G49 (G49.1)	08	Tool length compensation cancel
G44.9	27	Spindle unit compensation
G49.9		Spindle unit compensation cancel
G50	11	Scaling cancel
G51		Scaling
G50.1	22	Programmable mirror image cancel
G51.1		Programmable mirror image
G50.2	31	Polygon turning cancel
G51.2		Polygon turning
G50.4	00	Cancel synchronous control
G50.5		Cancel composite control
G50.6		Cancel superimposed control
G51.4		Start synchronous control
G51.5		Start composite control
G51.6		Start superimposed control

Table 3.1 (a) G code list

G code	Group	Function
G52	00	Local coordinate system setting
G53		Machine coordinate system setting
G53.1		Tool axis direction control
G53.6		Tool center point retention type tool axis direction control
G54 (G54.1)	14	Workpiece coordinate system 1 selection
G55		Workpiece coordinate system 2 selection
G56		Workpiece coordinate system 3 selection
G57		Workpiece coordinate system 4 selection
G58		Workpiece coordinate system 5 selection
G59		Workpiece coordinate system 6 selection
G54.4	33	Workpiece setting error compensation
G60	00	Single direction positioning
G61	15	Exact stop mode
G62		Automatic corner override
G63		Tapping mode
G64		Cutting mode
G65	00	Macro call
G66	12	Macro modal call A
G66.1		Macro modal call B
G67		Macro modal call A/B cancel
G68	16	Coordinate system rotation start or 3-dimensional coordinate conversion mode on
G69		Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off
G68.2		Tilted working plane command
G68.3		Tilted working plane command by tool axis direction
G68.4		Tilted working plane command (incremental multi-command)
G70.7	00	Finishing cycle
G71.7		Outer surface rough machining cycle
G72.7		End rough machining cycle
G73.7		Closed loop cutting cycle
G74.7		End cutting off cycle
G75.7		Outer or inner cutting off cycle
G76.7		Multiple threading cycle
G72.1		Figure copying (rotary copy)
G72.2		Figure copying (linear copy)
G73	09	Peck drilling cycle
G74		Left-handed tapping cycle
G75	01	Plunge grinding cycle
G76	09	Fine boring cycle
G77	01	Plunge direct sizing/grinding cycle
G78		Continuous-feed surface grinding cycle
G79		Intermittent-feed surface grinding cycle
G80	09	Canned cycle cancel Electronic gear box : synchronization cancellation
G80.4	34	Electronic gear box: synchronization cancellation
G81.4		Electronic gear box: synchronization start
G80.5	24	Electronic gear box 2 pair: synchronization cancellation
G81.5		Electronic gear box 2 pair: synchronization start
G81	09	Drilling cycle or spot boring cycle Electronic gear box : synchronization start
G81.1	00	Chopping function/High precision oscillation function

Table 3.1 (a) G code list

G code	Group	Function
G82	09	Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G84.2		Rigid tapping cycle (FS15 format)
G84.3		Left-handed rigid tapping cycle (FS15 format)
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring cycle
G90	03	Absolute programming
G91		Incremental programming
G91.1	00	Checking the maximum incremental amount specified
G92		Setting for workpiece coordinate system or clamp at maximum spindle speed
G92.1		Workpiece coordinate system preset
G93	05	Inverse time feed
G94		Feed per minute
G95		Feed per revolution
G96	13	Constant surface speed control
G97		Constant surface speed control cancel
G96.1	00	Spindle indexing execution (waiting for completion)
G96.2		Spindle indexing execution (not waiting for completion)
G96.3		Spindle indexing completion check
G96.4		SV speed control mode ON
G98	10	Canned cycle : return to initial level
G99		Canned cycle : return to R point level
G107	00	Cylindrical interpolation
G112	21	Polar coordinate interpolation mode
G113		Polar coordinate interpolation mode cancel
G160	20	In-feed control cancel
G161		In-feed control

3.2 G CODE LIST IN THE LATHE SYSTEM

T

Table 3.2 (b) G code list

G code system			Group	Function
A	B	C		
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01		Linear interpolation (Cutting feed)
G02	G02	G02		Circular interpolation CW or helical interpolation CW
G03	G03	G03		Circular interpolation CCW or helical interpolation CCW
G02.2	G02.2	G02.2		Involute interpolation CW
G02.3	G02.3	G02.3		Exponential interpolation CW
G02.4	G02.4	G02.4		3-dimensional coordinate system conversion CW
G03.2	G03.2	G03.2		Involute interpolation CCW
G03.3	G03.3	G03.3		Exponential interpolation CCW
G03.4	G03.4	G03.4		3-dimensional coordinate system conversion CCW
G04	G04	G04	00	Dwell
G05	G05	G05		AI contour control (command compatible with high precision contour control), High-speed cycle machining, High-speed binary program operation
G05.1	G05.1	G05.1		AI contour control / Nano smoothing / Smooth interpolation
G05.4	G05.4	G05.4		HRV3, 4 on/off
G06.2	G06.2	G06.2	01	NURBS interpolation
G07	G07	G07	00	Hypothetical axis interpolation
G07.1 (G107)	G07.1 (G107)	G07.1 (G107)		Cylindrical interpolation
G08	G08	G08		Advanced preview control
G09	G09	G09		Exact stop
G10	G10	G10		Programmable data input
G10.6	G10.6	G10.6		Tool retract and recover
G10.9	G10.9	G10.9		Programmable switching of diameter/radius specification
G11	G11	G11		Programmable data input mode cancel
G12.1 (G112)	G12.1 (G112)	G12.1 (G112)	21	Polar coordinate interpolation mode
G13.1 (G113)	G13.1 (G113)	G13.1 (G113)		Polar coordinate interpolation cancel mode
G17	G17	G17	16	XpYp plane selection
G17.1	G17.1	G17.1		Plane conversion function
G18	G18	G18		ZpXp plane selection
G19	G19	G19		YpZp plane selection
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
G22	G22	G22	09	Stored stroke check function on
G23	G23	G23		Stored stroke check function off
G25	G25	G25	08	Spindle speed fluctuation detection off
G26	G26	G26		Spindle speed fluctuation detection on
G27	G27	G27	00	Reference position return check
G28	G28	G28		Return to reference position
G28.2	G28.2	G28.2		In-position check disable reference position return
G29	G29	G29		Movement from reference position
G30	G30	G30		2nd, 3rd and 4th reference position return
G30.1	G30.1	G30.1		Floating reference point return

Table 3.2 (b) G code list

G code system			Group	Function
A	B	C		
G30.2	G30.2	G30.2	00	In-position check disable 2nd, 3rd, or 4th reference position return
G31	G31	G31		Skip function
G31.8	G31.8	G31.8		EGB-axis skip
G32	G33	G33	01	Threading
G34	G34	G34		Variable lead threading
G35	G35	G35		Circular threading CW
G36	G36	G36		Circular threading CCW (When bit 3 (G36) of parameter No. 3405 is set to 1) or Automatic tool offset (X axis) (When bit 3 (G36) of parameter No. 3405 is set to 0)
G37	G37	G37		Automatic tool offset (Z axis) (When bit 3 (G36) of parameter No. 3405 is set to 0)
G37.1	G37.1	G37.1		Automatic tool offset (X axis) (When bit 3 (G36) of parameter No. 3405 is set to 1)
G37.2	G37.2	G37.2		Automatic tool offset (Z axis) (When bit 3 (G36) of parameter No. 3405 is set to 1)
G38	G38	G38		Tool radius/tool nose radius compensation: with vector held
G39	G39	G39		Tool radius/tool nose radius compensation: corner rounding interpolation
G40	G40	G40	07	Tool radius/tool nose radius compensation : cancel
G41	G41	G41		Tool radius/tool nose radius compensation : left
G42	G42	G42		Tool radius/tool nose radius compensation : right
G41.2	G41.2	G41.2		3-dimensional cutter compensation : left (type 1)
G41.3	G41.3	G41.3		3-dimensional cutter compensation : (leading edge offset)
G41.4	G41.4	G41.4		3-dimensional cutter compensation : left (type 1) (FS16i-compatible command)
G41.5	G41.5	G41.5		3-dimensional cutter compensation : left (type 1) (FS16i-compatible command)
G41.6	G41.6	G41.6		3-dimensional cutter compensation : left (type 2)
G42.2	G42.2	G42.2		3-dimensional cutter compensation : right (type 1)
G42.4	G42.4	G42.4		3-dimensional cutter compensation : right (type 1) (FS16i-compatible command)
G42.5	G42.5	G42.5		3-dimensional cutter compensation : right (type 1) (FS16i-compatible command)
G42.6	G42.6	G42.6		3-dimensional cutter compensation : right (type 2)
G40.1	G40.1	G40.1	19	Normal direction control cancel mode
G41.1	G41.1	G41.1		Normal direction control left on
G42 .1	G42 .1	G42 .1		Normal direction control right on
G43	G43	G43	23	Tool length compensation + (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G44	G44	G44		Tool length compensation - (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G43.1	G43.1	G43.1		Tool length compensation in tool axis direction (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G43.4	G43.4	G43.4		Tool center point control (type 1) (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G43.5	G43.5	G43.5		Tool center point control (type 2) (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G43.7 (G44.7)	G43.7 (G44.7)	G43.7 (G44.7)		Tool offset (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G44.1	G44.1	G44.1		Tool offset conversion (Bit 3 (TCT) of parameter No. 5040 must be "1".)

Table 3.2 (b) G code list

G code system			Group	Function
A	B	C		
G49 (G49.1)	G49 (G49.1)	G49 (G49.1)	23	Tool length compensation cancel (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G50	G92	G92	00	Coordinate system setting or max spindle speed clamp
G50.3	G92.1	G92.1		Workpiece coordinate system preset
-	G50	G50	18	Scaling cancel
-	G51	G51		Scaling
G50.1	G50.1	G50.1	22	Programmable mirror image cancel
G51.1	G51.1	G51.1		Programmable mirror image
G50.2 (G250)	G50.2 (G250)	G50.2 (G250)	20	Polygon turning cancel
G51.2 (G251)	G51.2 (G251)	G51.2 (G251)		Polygon turning
G50.4	G50.4	G50.4	00	Cancel synchronous control
G50.5	G50.5	G50.5		Cancel composite control
G50.6	G50.6	G50.6		Cancel superimposed control
G51.4	G51.4	G51.4		Start synchronous control
G51.5	G51.5	G51.5		Start composite control
G51.6	G51.6	G51.6		Start superimposed control
G52	G52	G52		Local coordinate system setting
G53	G53	G53		Machine coordinate system setting
G53.1	G53.1	G53.1		Tool axis direction control
G53.6	G53.6	G53.6		Tool center point retention type tool axis direction control
G54 (G54.1)	G54 (G54.1)	G54 (G54.1)	14	Workpiece coordinate system 1 selection
G55	G55	G55		Workpiece coordinate system 2 selection
G56	G56	G56		Workpiece coordinate system 3 selection
G57	G57	G57		Workpiece coordinate system 4 selection
G58	G58	G58		Workpiece coordinate system 5 selection
G59	G59	G59		Workpiece coordinate system 6 selection
G54.4	G54.4	G54.4	26	Workpiece setting error compensation
G60	G60	G60	00	Single direction positioning
G61	G61	G61	15	Exact stop mode
G62	G62	G62		Automatic corner override mode
G63	G63	G63		Tapping mode
G64	G64	G64		Cutting mode
G65	G65	G65	00	Macro call
G66	G66	G66	12	Macro modal call A
G66.1	G66.1	G66.1		Macro modal call B
G67	G67	G67		Macro modal call A/B cancel
G68	G68	G68	04	Mirror image on for double turret or balance cutting mode
G68.1	G68.1	G68.1	17	Coordinate system rotation start or 3-dimensional coordinate system conversion mode on
G68.2	G68.2	G68.2		Tilted working plane command
G68.3	G68.3	G68.3		Tilted working plane command by tool axis direction
G68.4	G68.4	G68.4		Tilted working plane command (incremental multi-command)
G69	G69	G69	04	Mirror image off for double turret or balance cutting mode cancel
G69.1	G69.1	G69.1	17	Coordinate system rotation cancel or 3-dimensional coordinate system conversion mode off
G70	G70	G72	00	Finishing cycle
G71	G71	G73		Stock removal in turning
G72	G72	G74		Stock removal in facing

Table 3.2 (b) G code list

G code system			Group	Function
A	B	C		
G73	G73	G75	00	Pattern repeating cycle
G74	G74	G76		End face peck drilling cycle
G75	G75	G77		Outer diameter/internal diameter drilling cycle
G76	G76	G78		Multiple-thread cutting cycle
G72.1	G72.1	G72.1		Figure copying (rotational copying)
G72.2	G72.2	G72.2		Figure copying (linear copying)
G71	G71	G72	01	Traverse grinding cycle
G72	G72	G73		Traverse direct sizing/grinding cycle
G73	G73	G74		Oscillation grinding cycle
G74	G74	G75		Oscillation direct sizing/grinding cycle
G80	G80	G80	10	Canned cycle cancel for drilling Electronic gear box : synchronization cancellation
G81.1	G81.1	G81.1	00	Chopping function/High precision oscillation function
G80.4	G80.4	G80.4	28	Electronic gear box: synchronization cancellation
G81.4	G81.4	G81.4		Electronic gear box: synchronization start
G80.5	G80.5	G80.5	27	Electronic gear box 2 pair: synchronization cancellation
G81.5	G81.5	G81.5		Electronic gear box 2 pair: synchronization start
G81	G81	G81	10	Spot drilling (FS15-T format) Electronic gear box : synchronization start
G82	G82	G82		Counter boring (FS15-T format)
G83	G83	G83		Cycle for face drilling
G83.1	G83.1	G83.1		High-speed peck drilling cycle (FS15-T format)
G83.5	G83.5	G83.5		High-speed peck drilling cycle
G83.6	G83.6	G83.6		Peck drilling cycle
G84	G84	G84		Cycle for face tapping
G84.2	G84.2	G84.2		Rigid tapping cycle (FS15-T format)
G85	G85	G85		Cycle for face boring
G87	G87	G87		Cycle for side drilling
G87.5	G87.5	G87.5		High-speed peck drilling cycle
G87.6	G87.6	G87.6		Peck drilling cycle
G88	G88	G88		Cycle for side tapping
G89	G89	G89		Cycle for side boring
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Threading cycle
G94	G79	G24		End face turning cycle
G91.1	G91.1	G91.1	00	Maximum specified incremental amount check
G96	G96	G96	02	Constant surface speed control
G97	G97	G97		Constant surface speed control cancel
G96.1	G96.1	G96.1	00	Spindle indexing execution (waiting for completion)
G96.2	G96.2	G96.2		Spindle indexing execution (not waiting for completion)
G96.3	G96.3	G96.3		Spindle indexing completion check
G96.4	G96.4	G96.4		SV speed control mode ON
G93	G93	G93	05	Inverse time feed
G98	G94	G94		Feed per minute
G99	G95	G95		Feed per revolution
-	G90	G90	03	Absolute programming
-	G91	G91		Incremental programming
-	G98	G98	11	Canned cycle : return to initial level
-	G99	G99		Canned cycle : return to R point level

4 INTERPOLATION FUNCTIONS

Interpolation functions specify the way to make an axis movement (in other words, a movement of the tool with respect to the workpiece or table).

Chapter 4, "INTERPOLATION FUNCTIONS", consists of the following sections:

4.1 POSITIONING (G00) 40
4.2 SINGLE DIRECTION POSITIONING (G60) 41
4.3 LINEAR INTERPOLATION (G01) 44
4.4 CIRCULAR INTERPOLATION (G02, G03) 46
4.5 HELICAL INTERPOLATION (G02, G03) 50
4.6 HELICAL INTERPOLATION B (G02, G03) 51
4.7 SPIRAL INTERPOLATION, CONICAL INTERPOLATION (G02, G03) 52
4.8 POLAR COORDINATE INTERPOLATION (G12.1, G13.1) 58
4.9 CYLINDRICAL INTERPOLATION (G07.1) 66
4.10 CUTTING POINT INTERPOLATION FOR CYLINDRICAL INTERPOLATION (G07.1) 71
4.11 EXPONENTIAL INTERPOLATION (G02.3, G03.3) 81
4.12 SMOOTH INTERPOLATION (G05.1) 87
4.13 NANO SMOOTHING 92
4.14 NURBS INTERPOLATION (G06.2) 100
4.15 HYPOTHETICAL AXIS INTERPOLATION (G07) 108
4.16 VARIABLE LEAD THREADING (G34) 110
4.17 CIRCULAR THREADING (G35, G36) 111
4.18 SKIP FUNCTION (G31) 115
4.19 MULTI-STEP SKIP (G31) 117
4.20 HIGH-SPEED SKIP SIGNAL (G31) 117
4.21 SKIP POSITION MACRO VARIABLE IMPROVEMENT 118
4.22 CONTINUOUS HIGH-SPEED SKIP FUNCTION 118
4.23 TORQUE LIMIT SKIP 119
4.24 3-DIMENSIONAL CIRCULAR INTERPOLATION 122

4.1 POSITIONING (G00)

The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental programming at a rapid traverse rate.
In the absolute programming, coordinate value of the end point is programmed.
In the incremental programming the distance the tool moves is programmed.

Format

G00 IP_ ;

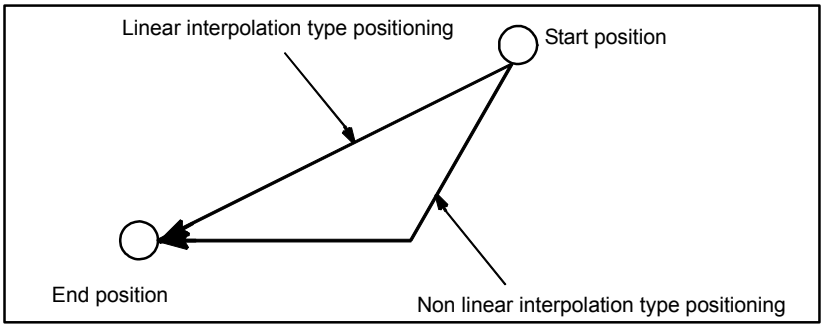
IP_ : For an absolute programming, the coordinates of an end point, and for an incremental programming, the distance the tool moves.

Explanation

Either of the following tool paths can be selected according to bit 1 (LRP) of parameter No. 1401.

- Nonlinear interpolation type positioning
  The tool is positioned with the rapid traverse rate for each axis separately. The tool path is normally straight.

- Linear interpolation type positioning
  The tool is positioned within the shortest possible time at a speed that is not more than the rapid traverse rate for each axis.



The rapid traverse rate in G00 command is set to the parameter No. 1420 for each axis independently by the machine tool builder. In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block. Execution proceeds to the next block after confirming the in-position.
"In-position " means that the feed motor is within the specified range.
This range is determined by the machine tool builder by setting to parameter No. 1826.

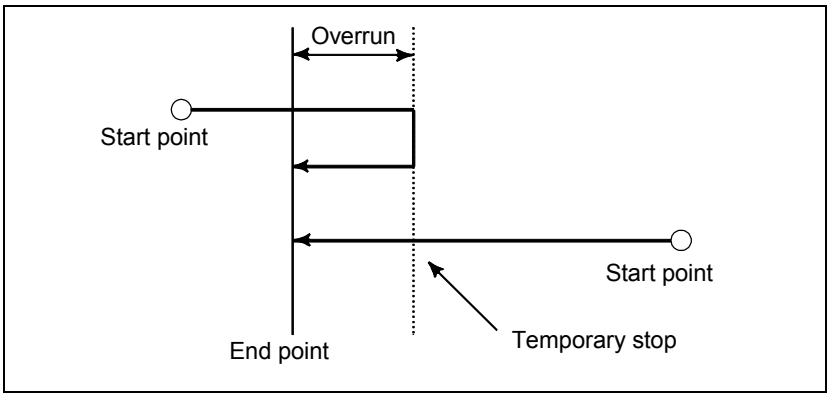
Limitation

The rapid traverse rate cannot be specified in the address F.
Even if linear interpolation type positioning is specified, nonlinear type interpolation positioning is used in the following cases. Therefore, be careful to ensure that the tool does not foul the workpiece.

- G28 specifying positioning between the reference and intermediate positions.
- G53

4.2 SINGLE DIRECTION POSITIONING (G60)

For accurate positioning without play of the machine (backlash), final positioning from one direction is available.



Format

G60 IP_ ;

IP_ : For an absolute programming, the coordinates of an end point, and for an incremental programming, the distance the tool moves.

Explanation

An overrun and a positioning direction are set by the parameter No. 5440. Even when a commanded positioning direction coincides with that set by the parameter, the tool stops once before the end point.

G60, which is a one-shot G-code, can be used as a modal G-code in group 01 by setting 1 to the bit 0 (MDL) of parameter No. 5431.
This setting can eliminate specifying a G60 command for every block. Other specifications are the same as those for a one-shot G60 command. When a one-shot G code is specified in the single direction positioning mode, the one-shot G command is effective like G codes in group 01.

(Example)

When one-shot G60 commands are used.
G90;
G60 X0Y0; }
G60 X100; } Single direction positioning
G60 Y100; }
G04 X10;
G00 X0Y0;
When modal G60 command is used.
G90G60; Single direction positioning mode start
X0Y0; }
X100; } Single direction positioning
Y100; }
G04X10;
G00X0 Y0; Single direction positioning mode cancel

- Overview of operation

- **In the case of positioning of non-linear interpolation type (bit 1 (LRP) of parameter No. 1401 = 0)**
  As shown below (Fig. 4.2 (a)), single direction positioning is performed independently along each axis.

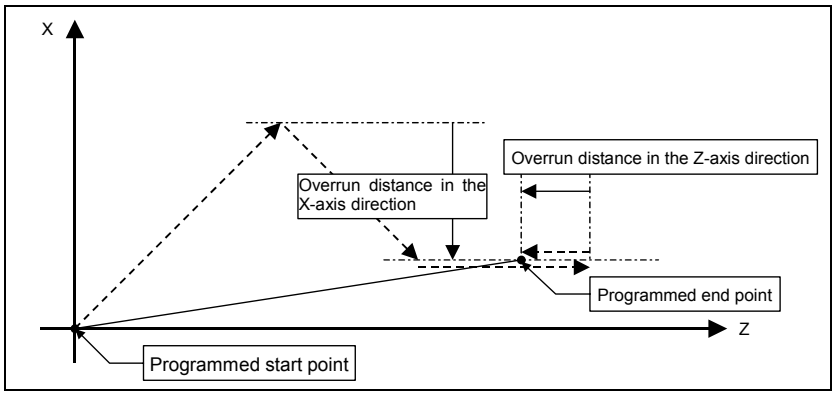


Fig. 4.2 (a)

- **In the case of positioning of linear interpolation type (bit 1 (LRP) of parameter No. 1401 = 1)**
  Positioning of interpolation type is performed until the tool once stops before or after a specified end point. Then, the tool is positioned independently along each axis until the end point is reached.

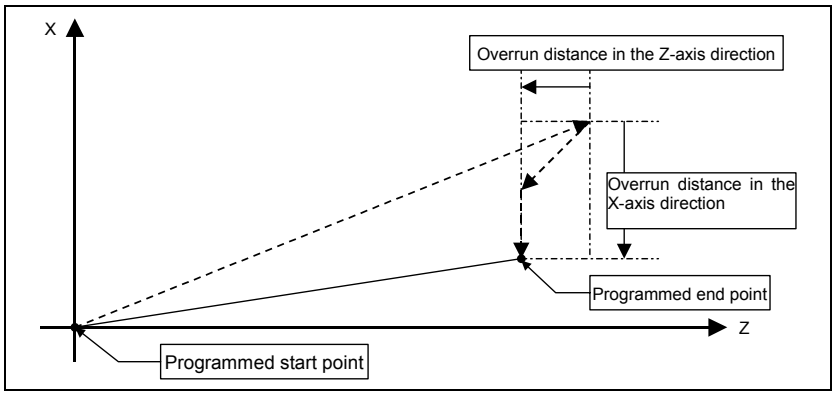


Fig. 4.2 (b)

Limitation

- Single direction positioning is not performed along an axis for which no overrun distance is set in parameter No. 5440.
- Single direction positioning is not performed along an axis for which travel distance 0 is specified.
- The mirror image function is not applied in a parameter-set direction. Even in the mirror image mode, the direction of single direction positioning remains unchanged. If positioning of linear interpolation type is used, and the state of mirror image when a single direction positioning block is looked ahead differs from the state of mirror image when the execution of the block is started, an alarm is issued. When switching mirror image in the middle of a program, disable looking ahead by specifying a non-buffering M code. Then, switch mirror image when there is no look-ahead block.
- In the cylindrical interpolation mode (G07.1), single direction positioning cannot be used.
- In the polar coordinate interpolation mode (G12.1), single direction positioning cannot be used.
- When specifying single direction positioning on a machine that uses arbitrary angular axis control, first position the angular axis then specify the positioning of the Cartesian axis. If the reverse specification order is used, or the angular axis and Cartesian axis are specified in the same block, an incorrect positioning direction can result.
- In positioning at a restart position by program restart function, single direction positioning is not performed.

M

- During canned cycle for drilling, no single direction positioning is effected in drilling axis.
- The single direction positioning does not apply to the shift motion in the canned cycles of G76 and G87.

T

- The G-code for single direction positioning is always G60, if G-code system is A or B or C in all case.
- The single direction positioning can not be commanded during the multiple repetitive cycle (G70-G76).
- No single direction positioning is effected in the drilling or patting axis, during canned cycle for drilling (G83-G89) and the rigid tapping (G84, G88). However, it can be commanded for positioning.
- The single direction positioning can not be commanded during the canned cycle (G90, G92, G94).
- During the single direction positioning mode (G60), the following G-code can not be commanded. G07.1, G12.1, G70-G76, G90-G94.

4.3 LINEAR INTERPOLATION (G01)

Tools can move along a line.

Format

G01 IP_ **F_ ;**

IP_ : For an absolute programming, the coordinates of an end point, and for an incremental programming, the distance the tool moves.
F_ : Speed of tool feed (Feedrate)

Explanation

A tools move along a line to the specified position at the feedrate specified in F.
The feedrate specified in F is effective until a new value is specified. It need not be specified for each block.
The feedrate commanded by the F code is measured along the tool path. If the F code is not commanded, the feedrate is regarded as zero.
The feedrate of each axis direction is as follows.

G01 αα ββ γγ ζζ Ff ;

Feed rate of α axis direction : $F\alpha = \frac{\alpha}{L} \times f$

Feed rate of β axis direction : $F\beta = \frac{\beta}{L} \times f$

Feed rate of γ axis direction : $F\gamma = \frac{\gamma}{L} \times f$

Feed rate of ζ axis direction : $F\zeta = \frac{\zeta}{L} \times f$

$L = \sqrt{\alpha^2 + \beta^2 + \gamma^2 + \zeta^2}$

The feedrate of the rotary axis is commanded in the unit of deg/min (the unit is decimal point position).
When the straight line axis α (such as X, Y, or Z) and the rotating axis β (such as A, B, or C) are linearly interpolated, the feedrate is that in which the tangential feedrate in the α and β cartesian coordinate system is commanded by F (mm/min).
β-axis feedrate is obtained ; at first, the time required for distribution is calculated by using the above formula, then the β-axis feedrate unit is changed to deg/min.

A calculation example is as follows.
G91 G01 X20.0B40.0 F300.0 ;
This changes the unit of the C axis from 40.0 deg to 40mm with metric input.
The time required for distribution is calculated as follows:

$$\frac{\sqrt{20^2 + 40^2}}{300} \fallingdotseq 0.14907(mm)$$

The feedrate for the C axis is

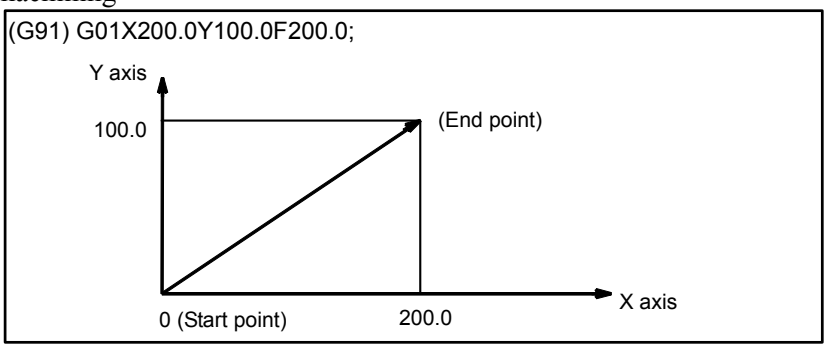
$$\frac{40}{0.14907} \fallingdotseq 268.3\,\mathrm{deg/\,min}$$

In simultaneous 3 axes control, the feedrate is calculated the same way as in 2 axes control.

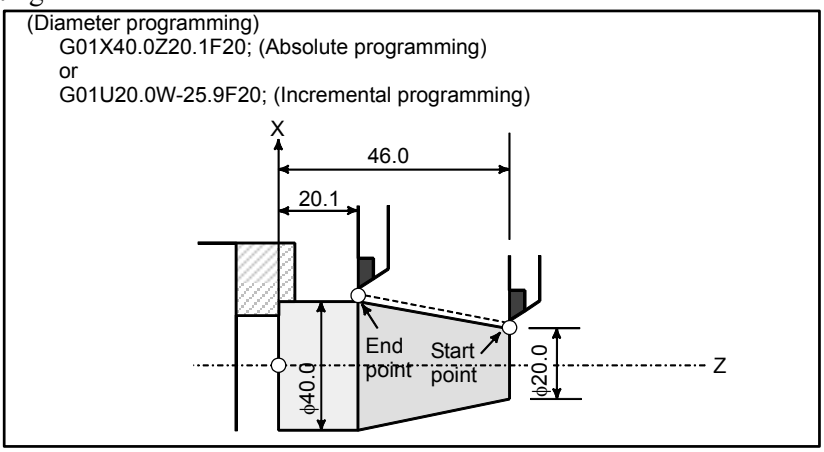
Example

- Linear interpolation

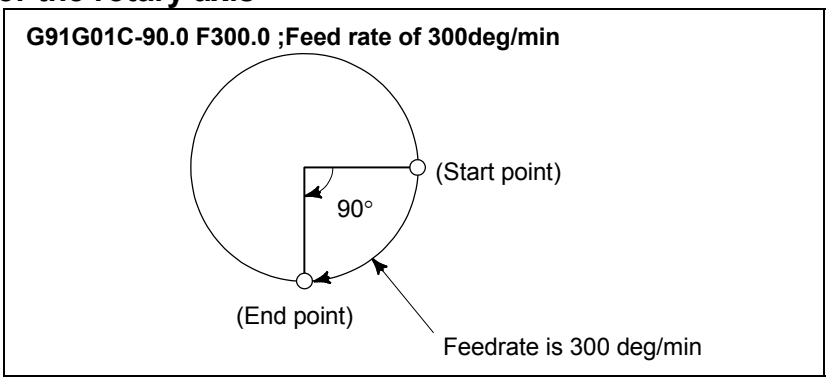
- For milling machining



- For lathe cutting



- Feedrate for the rotary axis



4.4 CIRCULAR INTERPOLATION (G02, G03)

The command below will move a tool along a circular arc.

Format

Arc in the XpYp plane

$$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Xp_\ Yp_ \begin{Bmatrix} I_\ J_ \\ R_ \end{Bmatrix} F_\ ;$$

Arc in the ZpXp plane

$$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Zp_\ Xp_ \begin{Bmatrix} I_\ K_ \\ R_ \end{Bmatrix} F_\ ;$$

Arc in the YpZp plane

$$G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Yp_\ Zp_ \begin{Bmatrix} J_\ K_ \\ R_ \end{Bmatrix} F_\ ;$$

Command	Description
G17	Specification of arc on XpYp plane
G18	Specification of arc on ZpXp plane
G19	Specification of arc on YpZp plane
G02	Circular Interpolation : Clockwise direction (CW)
G03	Circular Interpolation : Counterclockwise direction (CCW)
Xp_	Command values of X axis or its parallel axis (set by parameter No. 1022)
Yp	Command values of Y axis or its parallel axis (set by parameter No. 1022)
Zp_	Command values of Z axis or its parallel axis (set by parameter No. 1022)
I_	Xp axis distance from the start point to the center of an arc with sign
J_	Yp axis distance from the start point to the center of an arc with sign
K_	Zp axis distance from the start point to the center of an arc with sign
R_	Arc radius (with sign, radius value for lathe cutting)
F_	Feedrate along the arc

T

NOTE
The U-, V-, and W-axes can be used with G-codes B and C.

Explanation

- Direction of the circular interpolation

"Clockwise"(G02) and "counterclockwise"(G03) on the XpYp plane (ZpXp plane or YpZp plane) are defined when the XpYp plane is viewed in the positive-to-negative direction of the Zp axis (Yp axis or Xp axis, respectively) in the Cartesian coordinate system. See the figure below (Fig. 4.4 (a)).

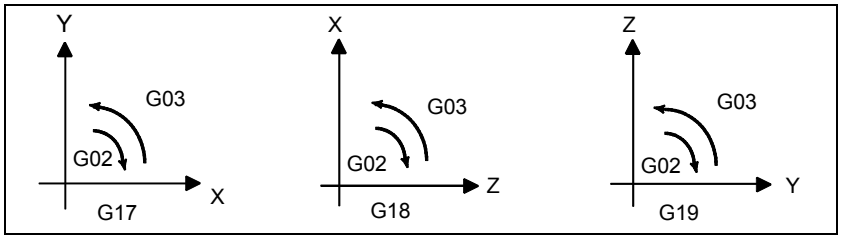


Fig. 4.4 (a)

- Distance moved on an arc

The end point of an arc is specified by address Xp, Yp or Zp, and is expressed as an absolute or incremental value according to G90 or G91. For the incremental value, the distance of the end point which is viewed from the start point of the arc is specified with sign.

- Distance from the start point to the center of arc

The arc center is specified by addresses I, J, and K for the Xp, Yp, and Zp axes, respectively. The numerical value following I, J, or K, however, is a vector component in which the arc center is seen from the start point, and is always specified as an incremental value irrespective of G90 and G91, as shown below (Fig. 4.4 (b)).

I, J, and K must be signed according to the direction.

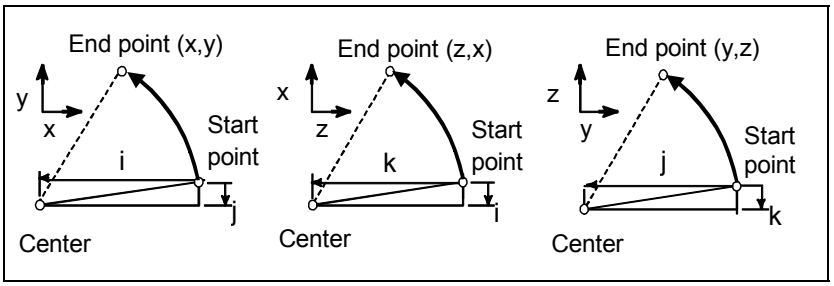


Fig. 4.4 (b)

I0, J0, and K0 can be omitted.

If the difference between the radius at the start point and that at the end point exceeds the permitted value in a parameter No.3410, an alarm PS0020 occurs.

- Command for a circle

When Xp, Yp, and Zp are omitted (the end point is the same as the start point) and the center is specified with I, J, and K, a 360° arc (circle) is specified.

G02 I_ ; Command for a circle

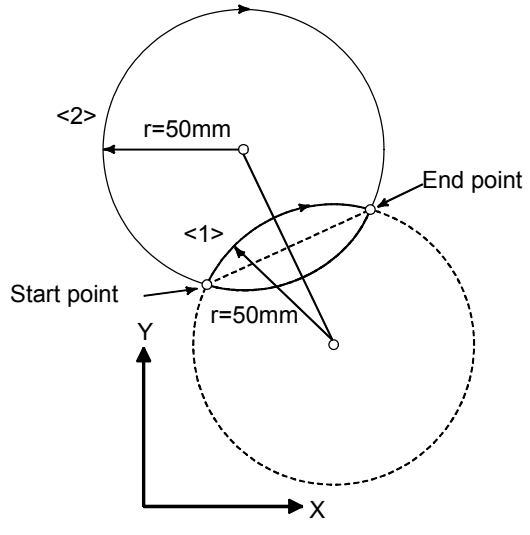
- Arc radius

The distance between an arc and the center of a circle that contains the arc can be specified using the radius, R, of the circle instead of I, J, and K.

In this case, one arc is less than 180°, and the other is more than 180° are considered. When an arc exceeding 180° is commanded, the radius must be specified with a negative value. If Xp, Yp, and Zp are all omitted, if the end point is located at the same position as the start point and when R is used, an arc of 0° is programmed

G02R_ ; (The cutter does not move.)

For arc <1> (less than 180°)
G91 G02 XP60.0 YP55.0 R50.0 F300.0 ;
For arc <2> (greater than 180°)
G91 G02 XP60.0 YP55.0 R-50.0 F300.0 ;



- Feedrate

The feedrate in circular interpolation is equal to the feedrate specified by the F code, and the feedrate along the arc (the tangential feedrate of the arc) is controlled to be the specified feedrate.
The error between the specified feedrate and the actual tool feedrate is ±2% or less. However, this feedrate is measured along the arc after the tool radius compensation is applied

Limitation

- Simultaneously specifying R with I, J, and K

If I, J, K, and R addresses are specified simultaneously, the arc specified by address R takes precedence and the other are ignored.

- Specifying an axis that is not contained in the specified plane

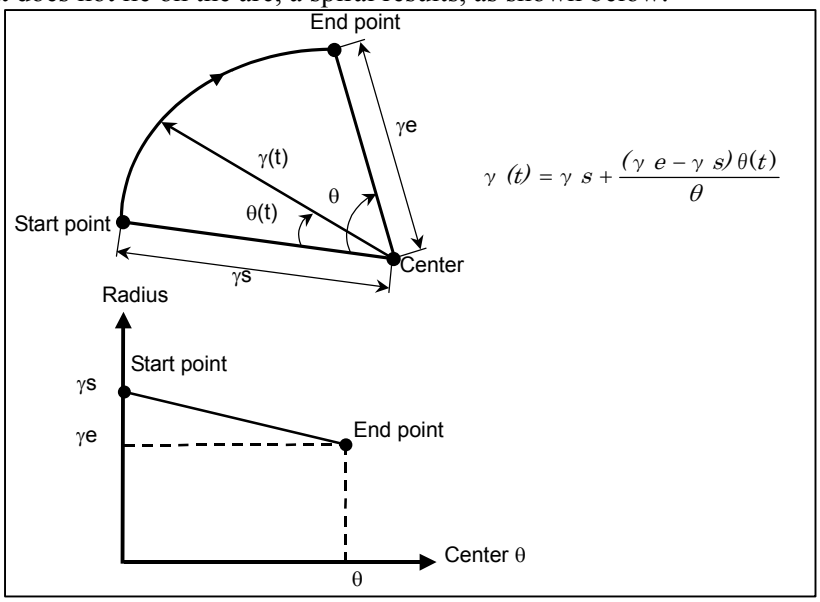
If an axis not comprising the specified plane is commanded, an alarm PS0021 occurs.
For example,
For milling machining:
If the X-axis and a U-axis parallel to the X-axis are specified when the XY plane is specified
For lathe cutting:
If the X-axis and a U-axis parallel to the X-axis are specified when the ZX plane is specified with G code system B or C

- Specifying a semicircle with R

When an arc having a center angle approaching 180° is specified, the calculated center coordinates may contain an error. In such a case, specify the center of the arc with I, J, and K.

- Difference in the radius between the start and end points

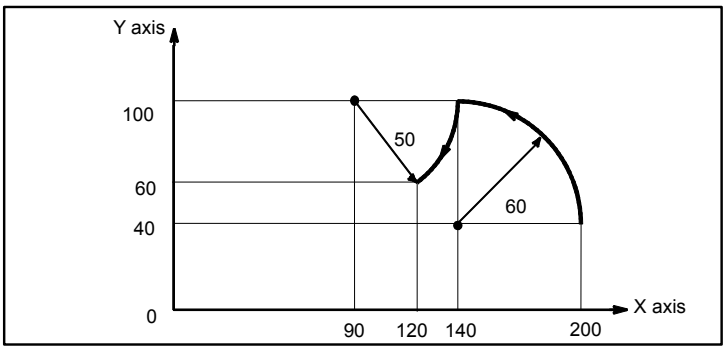
If the difference in the radius between the start and end points of the arc exceeds the value specified in parameter No. 3410, alarm PS0020 is generated.
When an end point does not lie on the arc, a spiral results, as shown below.



The arc radius changes linearly with the center angle θ(t). Spiral interpolation is performed using a circular command that specifies one arc radius for the start point and another arc radius for the end point. To use spiral interpolation, set a large value in parameter No. 3410, used to specify the limit on the arc radius error.

Example

M



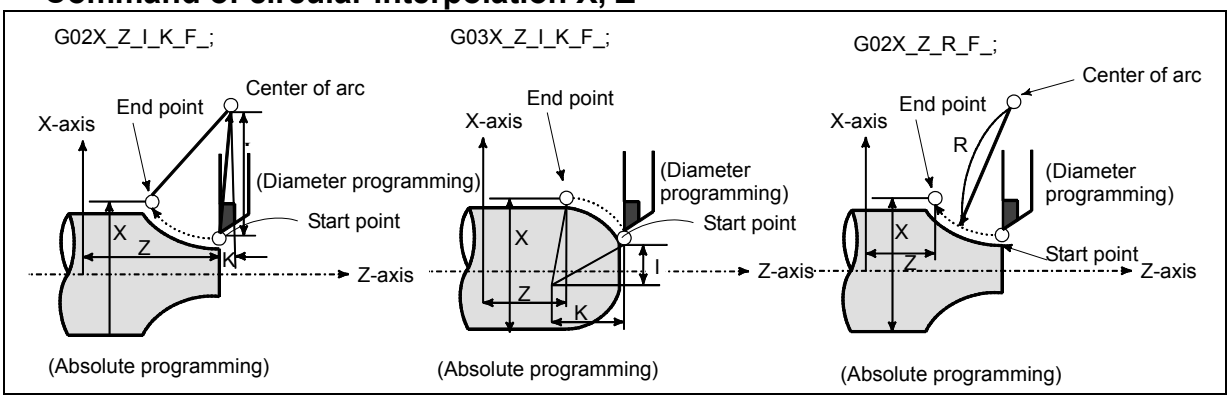
The above tool path can be programmed as follows;

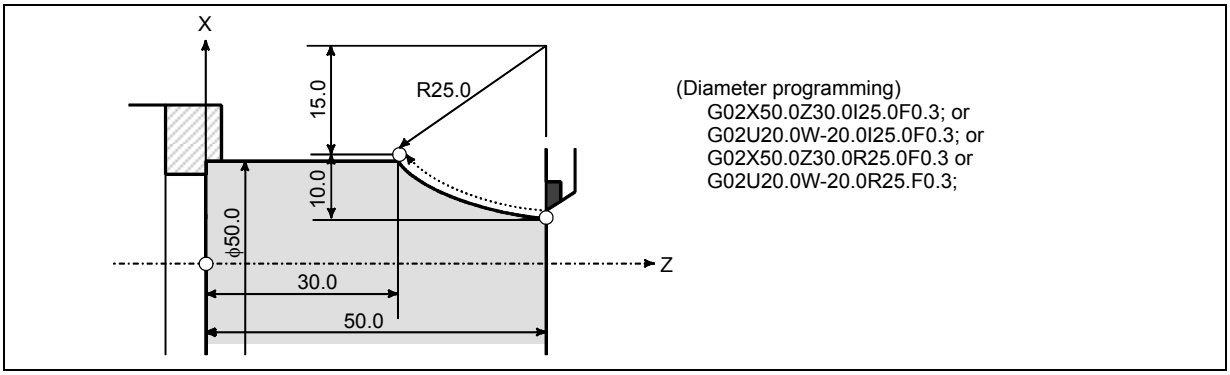
(1) In absolute programming
G92X200.0 Y40.0 Z0 ;
G90 G03 X140.0 Y100.0 R60.0 F300. ;
G02 X120.0 Y60.0 R50.0 ;
or
G92X200.0 Y40.0Z0 ;
G90 G03 X140.0 Y100.0I-60.0 F300. ;
G02 X120.0 Y60.0I-50.0 ;

(2) In incremental programming
G91 G03 X-60.0 Y60.0 R60.0 F300. ;
G02 X-20.0 Y-40.0 R50.0 ;
or
G91 G03 X-60.0 Y60.0 I-60.0 F300. ;
G02 X-20.0 Y-40.0 I-50.0 ;

T

- **Command of circular interpolation X, Z**





4.5 HELICAL INTERPOLATION (G02, G03)

Helical interpolation which moved helically is enabled by specifying up to two other axes which move synchronously with the circular interpolation by circular commands.

Format

Arc in the XpYp plane

$$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Xp_\ Yp_ \begin{Bmatrix} I_\ J_ \\ R_ \end{Bmatrix} \alpha_\ (\beta_)\ F_\ ;$$

Arc in the ZpXp plane

$$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Zp_\ Xp_ \begin{Bmatrix} K_\ I_ \\ R_ \end{Bmatrix} \alpha_\ (\beta_)\ F_\ ;$$

Arc in the YpZp plane

$$G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Yp_\ Zp_ \begin{Bmatrix} J_\ K_ \\ R_ \end{Bmatrix} \alpha_\ (\beta_)\ F_\ ;$$

α, β : Any one axis where circular interpolation is not applied.
Up to two other axes can be specified.

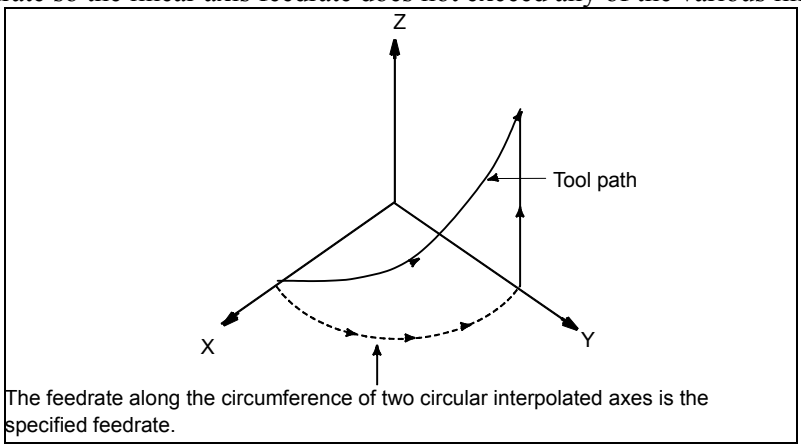
Explanation

A tangential velocity of an arc in a specified plane or a tangential velocity about the linear axis can be specified as the feedrate, depending on the setting of bit 5 (HTG) of parameter No.1403.

An F command specifies a feedrate along a circular arc, when HTG is specified to 0. Therefore, the feedrate of the linear axis is as follows:

$$F \times \frac{\text{Length of linear axis}}{\text{Length of circular arc}}$$

Determine the feedrate so the linear axis feedrate does not exceed any of the various limit values.



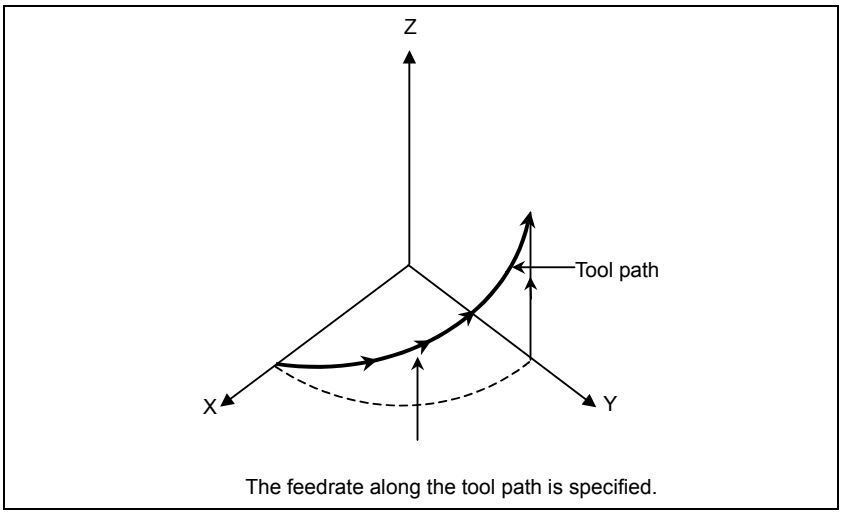
The feedrate along the circumference of two circular interpolated axes is the specified feedrate.

If HTG is set to 1, specify a feedrate along the tool path about the linear axis. Therefore, the tangential velocity of the arc is expressed as follows:

$$F \times \frac{\text{Length of arc}}{\sqrt{(\text{Length of arc})^2 + (\text{Length of linear axis})^2}}$$

The velocity along the linear axis is expressed as follows:

$$F \times \frac{\text{Length of linear axis}}{\sqrt{(\text{Length of arc})^2 + (\text{Length of linear axis})^2}}$$



The feedrate along the tool path is specified.

Limitation

- Tool radius/tool nose radius compensation is applied only for a circular arc.
- Tool offset and tool length compensation cannot be used in a block in which a helical interpolation is commanded.

4.6 HELICAL INTERPOLATION B (G02, G03)

The helical interpolation B function differs from the helical interpolation function just in that circular interpolation and a movement on four axes outside the specified plane can be simultaneously performed. For the restrictions and parameters, see the description of the helical interpolation function.

Format

Arc in the XpYp plane

$$\mathbf{G17}\ \begin{Bmatrix}\mathbf{G02}\\ \mathbf{G03}\end{Bmatrix}\ \mathbf{Xp_\ Yp_}\ \begin{Bmatrix}\mathbf{I_\ J_}\\ \mathbf{R_}\end{Bmatrix}\ \alpha_\ \beta_\ \gamma_\ \delta_\ \mathbf{F_}\ ;$$

Arc in the ZpXp plane

$$\mathbf{G18}\ \begin{Bmatrix}\mathbf{G02}\\ \mathbf{G03}\end{Bmatrix}\ \mathbf{Zp_\ Xp_}\ \begin{Bmatrix}\mathbf{K_\ I_}\\ \mathbf{R_}\end{Bmatrix}\ \alpha_\ \beta_\ \gamma_\ \delta_\ \mathbf{F_}\ ;$$

Arc in the YpZp plane

$$\mathbf{G19}\ \begin{Bmatrix}\mathbf{G02}\\ \mathbf{G03}\end{Bmatrix}\ \mathbf{Yp_\ Zp_}\ \begin{Bmatrix}\mathbf{J_\ K_}\\ \mathbf{R_}\end{Bmatrix}\ \alpha_\ \beta_\ \gamma_\ \delta_\ \mathbf{F_}\ ;$$

α, β, γ, δ : Any axis to which circular interpolation is not applied.
Up to four axes can be specified.

4.7 SPIRAL INTERPOLATION, CONICAL INTERPOLATION (G02, G03)

Spiral interpolation is enabled by specifying the circular interpolation command together with a desired number of revolutions or a desired increment (decrement) for the radius per revolution.
Conical interpolation is enabled by specifying the spiral interpolation command together with an additional axis of movement, as well as a desired increment (decrement) for the position along the additional axes per spiral revolution.

Format

- Spiral interpolation

XpYp plane

G17 {**G02**, **G03**} **X_Y_I_J_Q_L_F_;**

ZpXp plane

G18 {**G02**, **G03**} **Z_X_K_I_Q_L_F_;**

YpZp plane

G19 {**G02**, **G03**} **Y_Z_J_K_Q_L_F_;**

X, Y, Z : Coordinates of the end point
L : Number of revolutions (positive value without a decimal point) (*1)
Q : Radius increment or decrement per spiral revolution (*1, *2)
I, J, K : Signed distance from the start point to the center (same as the distance specified for circular interpolation)
F : Feedrate

(*1) Either the number of revolutions (L) or the radius increment or decrement (Q) can be omitted. When L is omitted, the number of revolutions is automatically calculated from the distance between the current position and the center, the position of the end point, and the radius increment or decrement. When Q is omitted, the radius increment or decrement is automatically calculated from the distance between the current position and the center, the position of the end point, and the number of revolutions. If both L and Q are specified but their values contradict, Q takes precedence. Generally, either L or Q should be specified. The L value must be a positive value without a decimal point. To specify four revolutions plus 90°, for example, round the number of revolutions up to five and specify L5.
(*2) The increment system for Q depends on the reference axis.

- Conical interpolation

XpYp plane

G17 {**G02** / **G03**} **X_ Y_ I_ J_ Z_ Q_ L_ F_ ;**

ZpXp plane

G18 {**G02** / **G03**} **Z_ X_ K_ I_ Y_ Q_ L_ F_ ;**

YpZp plane

G19 {**G02** / **G03**} **Y_ Z_ J_ K_ X_ Q_ L_ F_ ;**

X, Y, Z : Coordinates of the end point
L : Number of revolutions (positive value without a decimal point) (*1)
Q : Radius increment or decrement per spiral revolution (*1, *2)
I, J, K : Two of the three values represent a signed vector from the start point to the center. The remaining value is a height increment or decrement per spiral revolution in conical interpolation. (*1)
When the XpYp plane is selected:
The I and J values represent a signed vector from the start point to the center.
The K value represents a height increment or decrement per spiral revolution.
When the ZpXp plane is selected:
The K and I values represent a signed vector from the start point to the center.
The J value represents a height increment or decrement per spiral revolution.
When the YpZp plane is selected:
The J and K values represent a signed vector from the start point to the center.
The I value represents a height increment or decrement per spiral revolution.
F : Feedrate (The tangential velocity about the linear axis is specified.)

(*1) One of the height increment/decrement (I, J, K), radius increment/decrement (Q), and the number of revolutions (L) must be specified. The other two items can be omitted.

- Sample command for the XpYp plane

G18 {**G02** / **G03**} **X_ Y_ I_ J_ Z_** {**K_** / **Q_** / **L_**} **F_ ;**

If both L and Q are specified, but their values contradict, Q takes precedence. If both L and a height increment or decrement are specified, but their values contradict, the height increment or decrement takes precedence. If both Q and a height increment or decrement are specified, but their values contradict, Q takes precedence. The L value must be a positive value without a decimal point. To specify four revolutions plus 90°, for example, round the number of revolutions up to five and specify L5.

(*2) The increment system for Q depends on the reference axis.

Explanation

- Function of spiral interpolation

Spiral interpolation in the XY plane is defined as follows:

$$(X - X_0)^2 + (Y - Y_0)^2 = (R + Q')^2$$

X_0 : X coordinate of the center
Y_0 : Y coordinate of the center
R : Radius at the beginning of spiral interpolation
Q' : Variation in radius

When the programmed command is assigned to this function, the following expression is obtained:

$$(X - X_s - I)^2 + (Y - Y_s - J)^2 = (R + (L' + \frac{\theta}{360})Q)^2$$

where

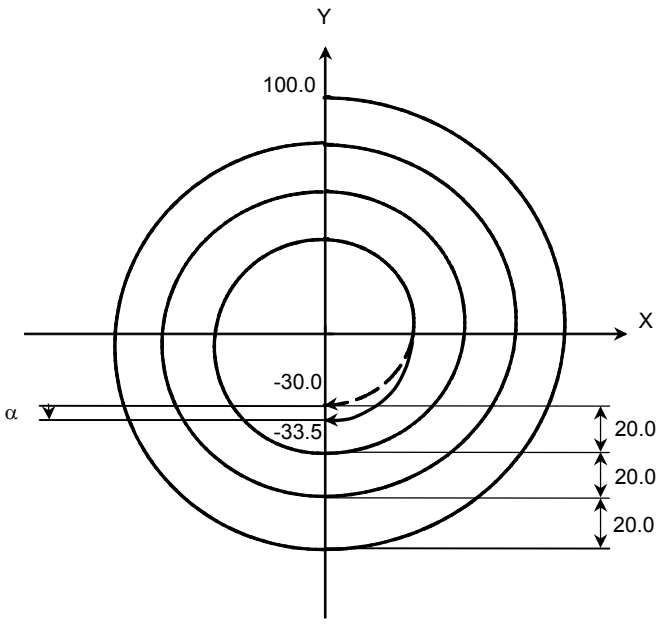
X_S	:	X coordinate of the start point
Y_S	:	Y coordinate of the start point
I	:	X coordinate of the vector from the start point to the center
J	:	Y coordinate of the vector from the start point to the center
R	:	Radius at the beginning of spiral interpolation
Q	:	Radius increment or decrement per spiral revolution
L'	:	(Current number of revolutions) - 1
θ	:	Angle between the start point and the current position (degrees)

- Controlled axes

For conical interpolation, two axes of a plane and two additional axes, that is, four axes in total, can be specified. A rotary axis can be specified as the additional axis.

- Difference between end points

If the difference between the programmed end point and the calculated end point of a spiral exceeds a value specified in parameter No. 3471 about any axis of a selected plane, an alarm PS5123 will be issued. If the difference between the programmed height and calculated height of the end point a cone exceeds a value specified in parameter No. 3471, an alarm PS5123 will be issued. The figure below (Fig. 4.7 (a)) illustrates details.



G90 G02 X0 Y-33.5 I0 J-100. { Q-20. / L4 } F300;

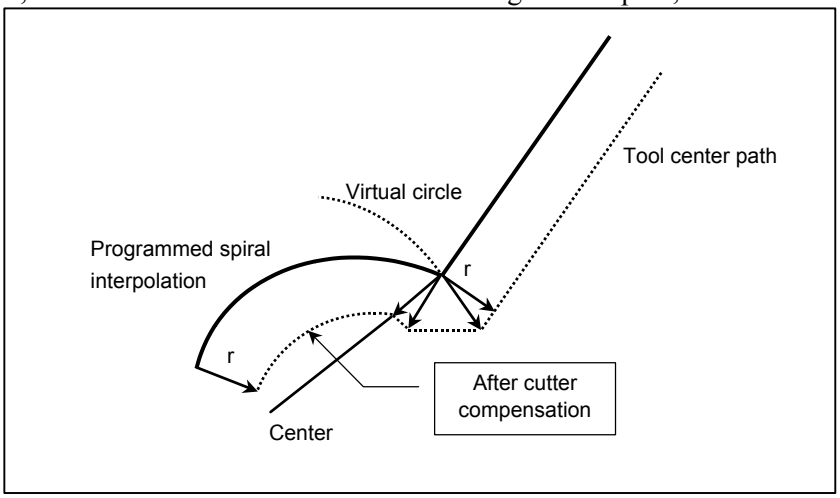
The coordinates of the programmed end point are (0, -33.5) while the coordinates of the calculated end point are (0, -30.0). A value greater than the difference (α: Tolerance) is specified in parameter No. 3471. If the end point is exceeded, an alarm PS5123 will be issued. The same is specified for the height of conical interpolation.

Fig. 4.7 (a)

- Tool radius compensation

M

The spiral or conical interpolation command can be programmed in tool radius compensation mode. This compensation is performed in the same way as described in "When it is exceptional" in "Tool Movement in Offset Mode" section. A virtual circle centered on the center of spiral interpolation is thought at the end of a block. The tool path is obtained by performing tool radius compensation along the virtual circle and the blocks before and after the spiral interpolation. When the end point of the block is at the center of the spiral interpolation, no virtual circle can be drawn. If drawing is attempted, an alarm PS5124 is issued.



- Actual cutting feedrate

A constant speed is maintained in spiral interpolation or conical interpolation. The angular velocity near the center, however, may increase because of the small radius of the spiral. This can be avoided by maintaining the angular velocity after the radius of the spiral reaches a value specified in parameter No. 3472. Consequently, the actual cutting feedrate decreases.
An example is illustrated below (Fig. 4.7 (b)).

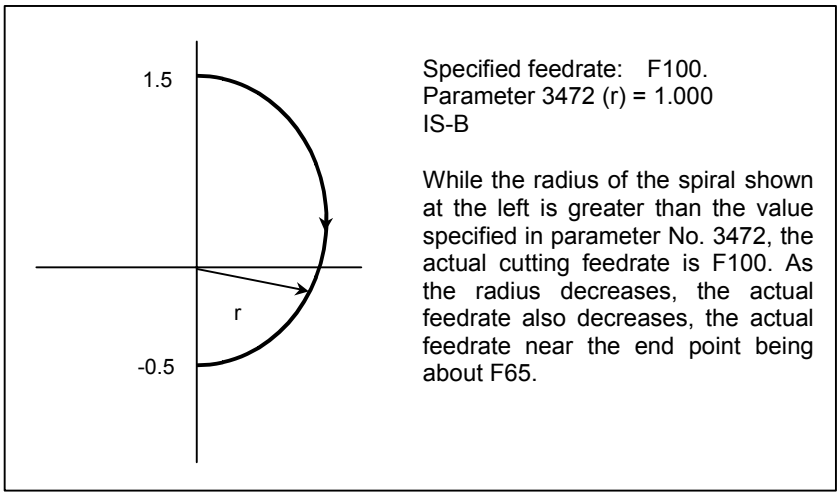


Fig. 4.7 (b)

- Deceleration by acceleration

During spiral interpolation, the function of deceleration by acceleration is enabled. The feedrate may decrease as the tool approaches the center of the spiral.

- Dry run

When the dry run signal is inverted from 0 to 1 or from 1 to 0 during movement along an axis, the movement is accelerated or decelerated to the desired speed without first reducing the speed to zero.

Limitation

- Radius

During spiral interpolation and conical interpolation, the addresses "C", "R", ",C", or ",R" cannot be specified.

- Feed functions

The functions of feed per rotation, inverse time feed, F command with one digit, and automatic corner override cannot be used.

- Retrace

A program including spiral or conical interpolation cannot be retraced.

- Polar coordinate interpolation, scaling, and normal direction control

Spiral interpolation and conical interpolation cannot be specified in these modes.

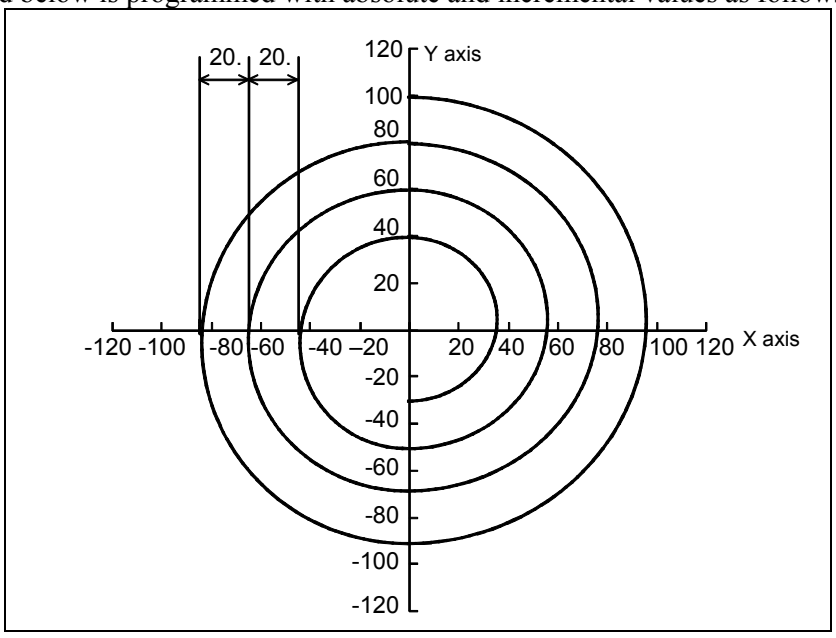
- Optional function

To use the conical interpolation function, the optional function for helical interpolation is also necessary.

Example

- Spiral interpolation

The path indicated below is programmed with absolute and incremental values as follows:



This sample path has the following values:

- Start point : (0, 100.0)
- End point (X, Y) : (0, -30.0)
- Distance to the center (I, J) : (0, -100.0)
- Radius increment or decrement (Q) : -20.0
- Number of revolutions (L) : 4

(1) With absolute values, the path is programmed as follows:

G90 G02 X0 Y-30.0 I0 J-100.0 $\left\{ \begin{matrix} \text{Q-20.0} \\ \text{L4} \end{matrix} \right\}$ F300.0 ;

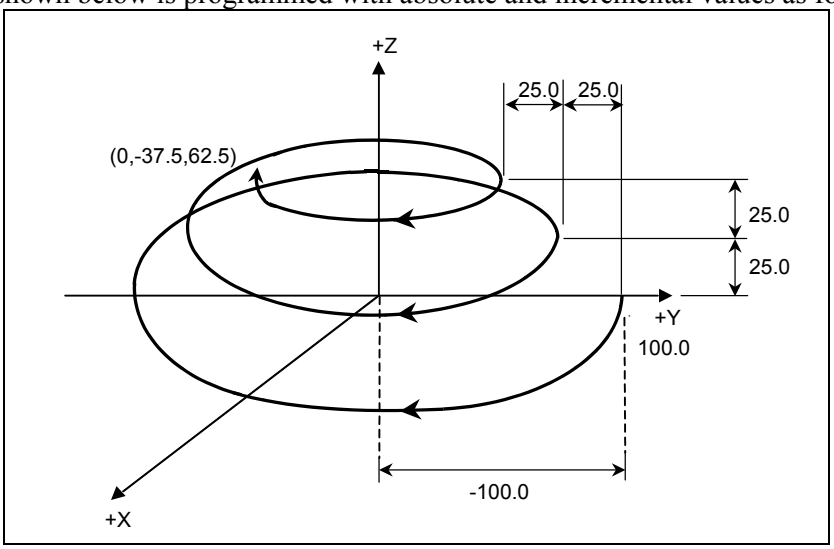
(2) With incremental values, the path is programmed as follows:

G91 G02 X0 Y-130.0 I0 J-100.0 { Q-20.0 / L4 } F300.0 ;

(Either the Q or L setting can be omitted.)

- Conical interpolation

The sample path shown below is programmed with absolute and incremental values as follows:



This sample path has the following values:

- Start point : (0, 100.0, 0)
- End point (X, Y, Z) : (0, -37.5, 62.5)
- Distance to the center (I, J) : (0, -100.0)
- Radius increment or decrement (Q) : -25.0
- Height increment or decrement (K) : 25.0
- Number of revolutions (L) : 3

(1) With absolute values, the path is programmed as follows:

G90 G02 X0 Y-37.5 Z62.5 I0 J-100.0 { K25.0 / Q-25.0 / L3 } F300.0 ;

(2) With incremental values, the path is programmed as follows:

G91 G02 X0 Y-137.5 Z62.5 I0 J-100.0 { K25.0 / Q-25.0 / L3 } F300.0 ;

(Either the Q or L setting can be omitted.)

4.8 POLAR COORDINATE INTERPOLATION (G12.1, G13.1)

Overview

Polar coordinate interpolation is a function that exercises contour control in converting a command programmed in a Cartesian coordinate system to the movement of a linear axis (movement of a tool) and the movement of a rotary axis (rotation of a workpiece). This function is useful for grinding a cam shaft.

Format

G12.1; Starts polar coordinate interpolation mode (enables polar coordinate interpolation).

Specify linear or circular interpolation using coordinates in a Cartesian coordinate system consisting of a linear axis and rotary axis (hypothetical axis).

G13.1; Polar coordinate interpolation mode is cancelled (for not performing polar coordinate interpolation).

Specify G12.1 and G13.1 in Separate Blocks.

Explanation

- Polar coordinate interpolation mode (G12.1)

The axes of polar coordinate interpolation (linear axis and rotary axis) should be specified in advance, with corresponding parameters. Specifying G12.1 places the system in the polar coordinate interpolation mode, and selects a plane (called the polar coordinate interpolation plane) formed by one linear axis and a hypothetical axis intersecting the linear axis at right angles. The linear axis is called the first axis of the plane, and the hypothetical axis is called the second axis of the plane. Polar coordinate interpolation is performed in this plane.

In the polar coordinate interpolation mode, both linear interpolation and circular interpolation can be specified by absolute or incremental programming.

Tool radius compensation can also be performed. The polar coordinate interpolation is performed for a path obtained after tool radius compensation.

The tangential velocity in the polar coordinate interpolation plane (Cartesian coordinate system) is specified as the feedrate, using F.

- Polar coordinate interpolation cancel mode (G13.1)

Specifying G13.1 cancels the polar coordinate interpolation mode.

- Polar coordinate interpolation plane

G12.1 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plane (Fig. 4.8 (a)). Polar coordinate interpolation is performed on this plane.

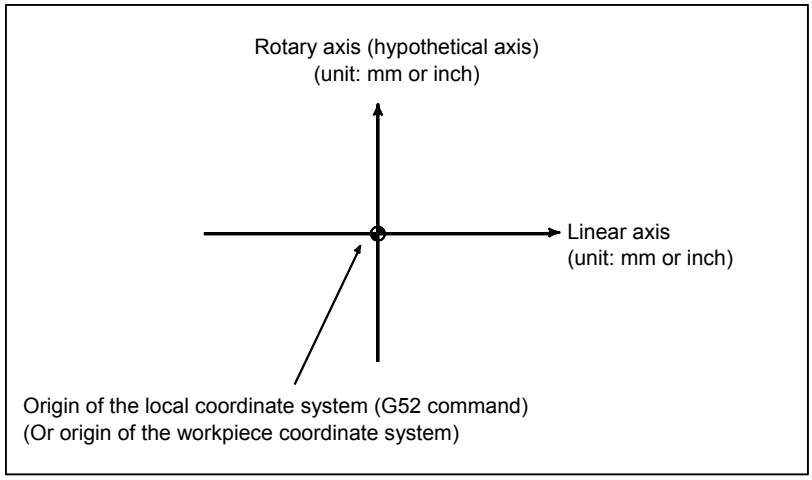


Fig. 4.8 (a) Polar coordinate interpolation plane

When the power is turned on or the system is reset, polar coordinate interpolation is canceled (G13.1).
The linear and rotation axes for polar coordinate interpolation must be set in parameters Nos. 5460 and 5461 beforehand.

> ⚠ **CAUTION**
> The plane used before G12.1 is specified (plane selected by G17, G18, or G19) is canceled. It is restored when G13.1 (canceling polar coordinate interpolation) is specified.
> When the system is reset, polar coordinate interpolation is canceled and the plane specified by G17, G18, or G19 is used.

- Distance moved and feedrate for polar coordinate interpolation

- The unit for coordinates on the hypothetical axis is the same as the unit for the linear axis (mm/inch).
  In the polar coordinate interpolation mode, program commands are specified with Cartesian coordinates on the polar coordinate interpolation plane. The axis address for the rotary axis is used as the axis address for the second axis (hypothetical axis) in the plane. Whether a diameter or radius is specified for the first axis in the plane is the same as for the rotary axis regardless of the specification for the first axis in the plane.
  The hypothetical axis is at coordinate 0 immediately after G12.1 is specified. Polar interpolation is started assuming the rotation angle of 0 for the position of the tool when G12.1 is specified.
  Example)

  When a value on the X-axis (linear axis) is input in millimeters
  G12.1;
  G01 X10. F1000. ; .....A 10-mm movement is made on the Cartesian coordinate system.
  C20. ;.....A 20-mm movement is made on the Cartesian coordinate system.
  G13.1;

  When a value on the X-axis (linear axis) is input in inches
  G12.1;
  G01 X10. F1000. ; .....A 10-inch movement is made on the Cartesian coordinate system.
  C20. ;.....A 20-inch movement is made on the Cartesian coordinate system.
  G13.1;

- The unit for the feedrate is mm/min or inch/min.
  Specify the feedrate as a speed (relative speed between the workpiece and tool) tangential to the polar coordinate interpolation plane (Cartesian coordinate system) using F.

- G codes which can be specified in the polar coordinate interpolation mode

G01 .....Linear interpolation
G02, G03 .....Circular interpolation
G02.2, G03.2 .....Involute interpolation
G04 .....Dwell, Exact stop
G40, G41, G42 .....Tool radius compensation (Polar coordinate interpolation is applied to the path after tool radius compensation.)
G65, G66, G67 .....Custom macro command
G90, G91 .....Absolute programming, incremental programming
G94, G95 .....Feed per minute, feed per revolution

- Circular interpolation in the polar coordinate plane

The addresses for specifying the radius of an arc for circular interpolation (G02 or G03) in the polar coordinate interpolation plane depend on the first axis in the plane (linear axis).

- I and J in the Xp-Yp plane when the linear axis is the X-axis or an axis parallel to the X-axis.
- J and K in the Yp-Zp plane when the linear axis is the Y-axis or an axis parallel to the Y-axis.
- K and I in the Zp-Xp plane when the linear axis is the Z-axis or an axis parallel to the Z-axis.

The radius of an arc can be specified also with an R command.

NOTE
In a lathe system, the parallel axes U, V, and W can be used in the G code system B or C.

- Movement along axes not in the polar coordinate interpolation plane in the polar coordinate interpolation mode

The tool moves along such axes normally, independent of polar coordinate interpolation.

- Current position display in the polar coordinate interpolation mode

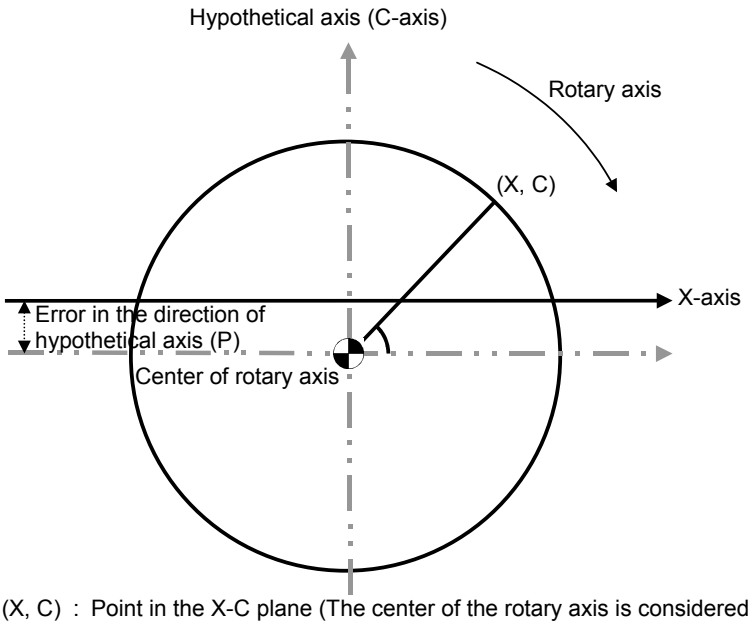
Actual coordinates are displayed. However, the remaining distance to move in a block is displayed based on the coordinates in the polar coordinate interpolation plane (Cartesian coordinates).

- Coordinate system for the polar coordinate interpolation

Basically, before G12.1 is specified, a local coordinate system (or workpiece coordinate system) where the center of the rotary axis is the origin of the coordinate system must be set.

- Compensation in the direction of the hypothetical axis in polar coordinate interpolation

If the first axis of the plane has an error from the center of the rotary axis in the hypothetical axis direction, in other words, if the rotary axis center is not on the X-axis, the hypothetical axis direction compensation function in the polar coordinate interpolation mode is used. With the function, the error is considered in polar coordinate interpolation. The amount of error is specified in parameter No. 5464.



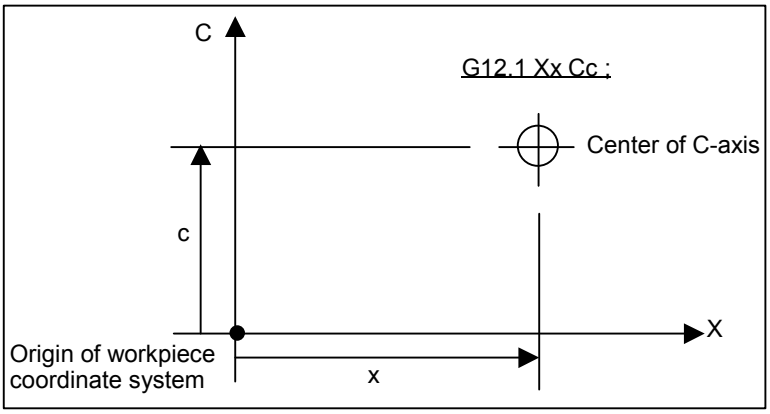
(X, C) : Point in the X-C plane (The center of the rotary axis is considered to be the origin of the X-C plane.)
X : X coordinate in the X-C plane
C : Hypothetical axis coordinate in the X-C plane
P : Error in the direction of the hypothetical axis (specified in parameter No. 5464)

- Shifting the coordinate system in polar coordinate interpolation

In the polar coordinate interpolation mode, the workpiece coordinate system can be shifted. The current position display function shows the position viewed from the workpiece coordinate system before the shift. The function to shift the coordinate system is enabled when bit 2 (PLS) of parameter No. 5450 is specified accordingly.

The shift can be specified in the polar coordinate interpolation mode, by specifying the position of the center of the rotary axis C (A, B) in the X-C (Y-A, Z-B) interpolation plane with reference to the origin of the workpiece coordinate system, in the following format.

```
G12.1 X_ C_ ;   (Polar coordinate interpolation for the X-axis and C-axis)
G12.1 Y_ A_ ;   (Polar coordinate interpolation for the Y-axis and A-axis)
G12.1 Z_ B_ ;   (Polar coordinate interpolation for the Z-axis and B-axis)
```



Limitation

- Changing the coordinate system during polar coordinate interpolation

In the G12.1 mode, the coordinate system must not be changed (G92, G52, G53, relative coordinate reset, G54 through G59, etc.).

- Tool radius/tool nose radius compensation

The polar coordinate interpolation mode (G12.1 or G13.1) cannot be started or terminated in the tool radius/tool nose radius compensation mode (G41 or G42). G12.1 or G13.1 must be specified in the tool radius/tool nose radius compensation canceled mode (G40).
For the tool radius/tool nose radius compensation canceled mode (G40) command, be sure to specify the polar coordinate axis to cancel the offset vector.

- Tool length offset command

Tool length offset must be specified in the polar coordinate interpolation cancel mode before G12.1 is specified. It cannot be specified in the polar coordinate interpolation mode. Furthermore, no offset values can be changed in the polar coordinate interpolation mode.

- Tool offset command

A tool offset must be specified before the G12.1 mode is set. No offset can be changed in the G12.1 mode.

- Program restart

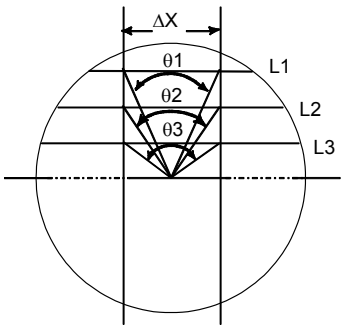
For a block in the G12.1 mode, the program and the block cannot be restarted.

- Cutting feedrate for the rotary axis

Polar coordinate interpolation converts the tool movement for a figure programmed in a Cartesian coordinate system to the tool movement in the rotary axis (C-axis) and the linear axis (X-axis). When the tool comes close to the center of the workpiece, the C-axis velocity component increases. If the maximum cutting feedrate for the C-axis (parameter No. 1430) is exceeded, the automatic feedrate override function and automatic speed clamp function are enabled.
If the maximum cutting feedrate for the X-axis is exceeded, the automatic feedrate override function and automatic speed clamp function are enabled.

⚠ WARNING

1 Consider lines L1, L2, and L3. ΔX is the distance the tool moves per time unit at the feedrate specified with address F in the Cartesian coordinate system. As the tool moves from L1 to L2 to L3, the angle at which the tool moves per time unit corresponding to ΔX in the Cartesian coordinate system increases from θ1 to θ2 to θ3. In other words, the C-axis component of the feedrate becomes larger as the tool moves closer to the center of the workpiece. The C component of the feedrate may exceed the maximum cutting feedrate for the C-axis because the tool movement in the Cartesian coordinate system has been converted to the tool movement for the C-axis and the X-axis.



L: Distance (in mm) between the tool center and workpiece center when the tool center is the nearest to the workpiece center

R: Maximum cutting feedrate (deg/min) of the C axis

Then, a speed specifiable with address F in polar coordinate interpolation can be given by the formula below. If the maximum cutting feedrate for the C-axis is exceeded, the automatic speed control function for polar coordinate interpolation automatically controls the feedrate.

$$F < L \times R \times \frac{\pi}{180} \quad \text{(mm/min)}$$

2 The following function cannot be used for the rotary axis of polar coordinate interpolation.

- Index table indexing function

- Automatic speed control for polar coordinate interpolation

If the velocity component of the rotary axis exceeds the maximum cutting feedrate in the polar coordinate interpolation mode, the speed is automatically controlled.

- Automatic override

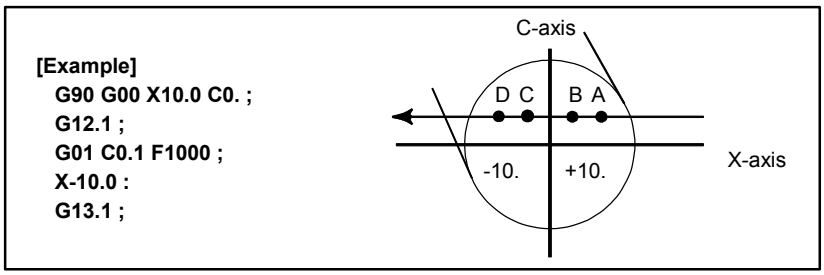
If the velocity component of the rotary axis exceeds the permissible velocity (maximum cutting feedrate multiplied by the permission factor specified in parameter No. 5463), the feedrate is automatically overridden as indicated below.

Override = (Permissible velocity) ÷ (Velocity component of rotary axis) × 100(%)

- Automatic speed clamp

If the velocity component of the rotary axis after automatic override still exceeds the maximum cutting feedrate, the speed of the rotary axis is automatically clamped. As a result, the velocity component of the rotary axis will not exceed the maximum cutting feedrate.

The automatic speed clamp function works only when the center of the tool is very close to the center of the rotary axis.



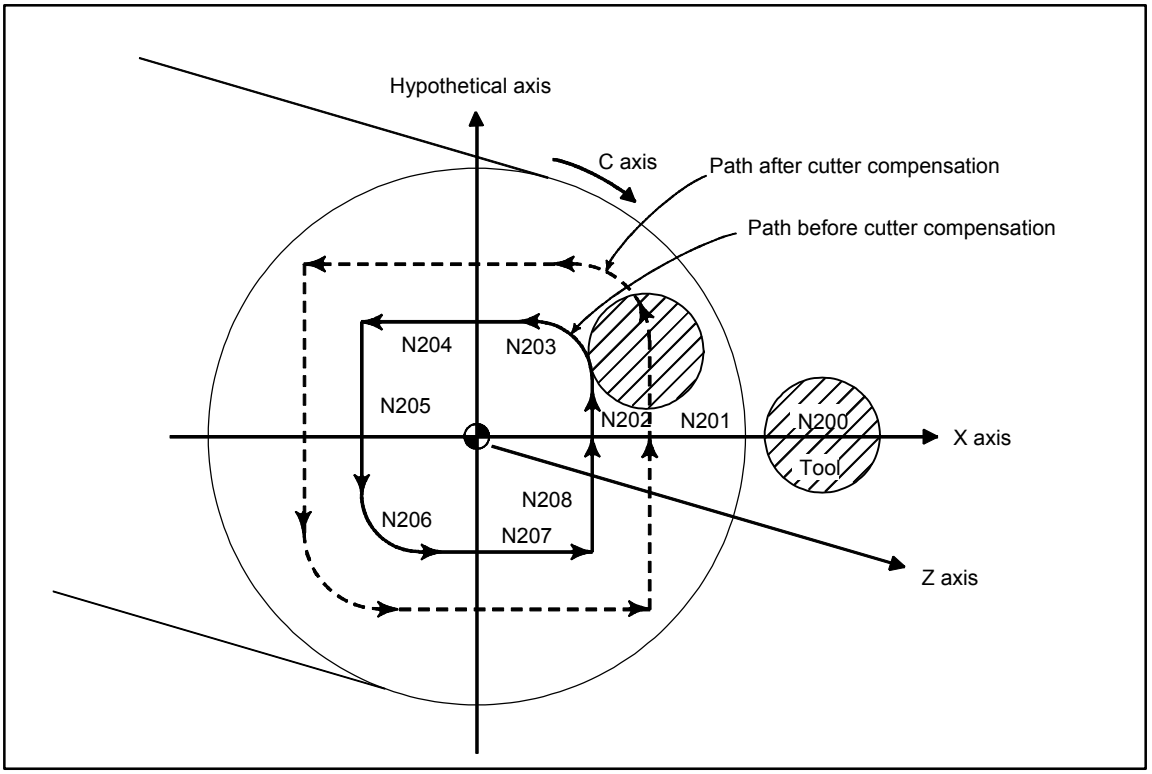
Automatic speed control for polar coordinate interpolation

Suppose that the maximum cutting feedrate of the rotary axis is 360 (3600 deg/min) and that the permission factor of automatic override for polar coordinate interpolation (parameter No. 5463) is 0 (90%). If the program indicated above is executed, the automatic override function starts working when the X coordinate becomes 2.273 (point A). The automatic speed clamp function starts working when the X coordinate becomes 0.524 (point B).
The minimum value of automatic override for this example is 3%. The automatic speed clamp function continues working until the X coordinate becomes -0.524 (point C). Then, the automatic override function works until the X coordinate becomes -2.273 (point D).
(The coordinates indicated above are the values in the Cartesian coordinate system.)

NOTE
1 While the automatic speed clamp function is working, the machine lock or interlock function may not be enabled immediately.
2 If a feed hold stop is made while the automatic speed clamp function is working, the automatic operation halt signal is output. However, the operation may not stop immediately.
3 The clamped speed may exceed the clamp value by a few percent.

Example

Sample program for polar coordinate interpolation in a Cartesian coordinate system consisting of the X-axis (a linear axis) and a hypothetical axis




```
O001;
   .
No10 T0101
   .
N0100 G90 G00 X60.0 C0 Z__;          Positioning to start point
N0200 G12.1;                         Start of polar coordinate interpolation
N0201 G42 G01 X20.0F__;
N0202 C10.0;
N0203 G03 X10.0 C20.0 R10.0;
N0204 G01 X-20.0;                    Geometry program
N0205 C-10.0;                        (program based on cartesian coordinates on
N0206 G03 X-10.0 C-20.0 I10.0 J0;    X axis-hypothetical axis plane)
N0207 G01 X20.0;
N0208 C0;
N0209 G40 X60.0;
N0210 G13.1;                         Cancellation of polar coordinate interpolation
N0300 Z__;
N0400 X__C__;
   .
N0900M30;
```

4.9 CYLINDRICAL INTERPOLATION (G07.1)

4.9.1 Cylindrical Interpolation

In cylindrical interpolation function, the amount of movement of a rotary axis specified by angle is converted to the amount of movement on the circumference to allow linear interpolation and circular interpolation with another axis.
Since programming is enabled with the cylinder side face expanded, programs such as a program for grooving cylindrical cams can be created very easily.

Format

G07.1 IP r; Starts the cylindrical interpolation mode (enables cylindrical interpolation).
:
:
:
G07.1 IP 0; The cylindrical interpolation mode is cancelled.
IP : An address for the rotary axis
r : The radius of the workpiece
Specify G07.1 IPr; and G07.1 IP0; in separate blocks.
G107 can be used instead of G07.1.

NOTE
Only a positive value is effective as the radius of the workpiece. If a negative value is specified, alarm PS0175 is issued.

Explanation

- Plane selection (G17, G18, G19)

To specify a G code for plane selection, set the rotary axis in parameter No. 1022 as a linear axis that is one of the basic three axes of the basic coordinate system or an axis parallel to one of the basic axes. For example, when rotary axis C-axis is assumed to be parallel to the X-axis, specifying G17, axis address C, and Y at the same time can select a plane formed by the C-axis and Y-axis (the Xp-Yp plane).

T

NOTE
The U-, V-, and W-axes can be used with G-codes B and C.

- Feedrate

A feedrate specified in the cylindrical interpolation mode is the feedrate on the circumference.

- Circular interpolation (G02, G03)

Circular interpolation can be performed between the rotary axis set for cylindrical interpolation and another linear axis. Radius R is used in commands in the same way as described.
The unit for a radius is not degrees but millimeters (for metric input) or inches (for inch input).

<Example Circular interpolation between the Z axis and C axis>
For the C axis of parameter No.1022, 5 (axis parallel with the X axis) is to be set. In this case, the command for circular interpolation is
G18 Z_C_;
G02 (G03) Z_C_R_;

For the C axis of parameter No.1022, 6 (axis parallel with the Y axis) may be specified instead. In this case, however, the command for circular interpolation is

G19 C_Z_;
G02 (G03) Z_C_R_;

- Tool radius/tool nose radius compensation

To perform tool radius/tool nose radius compensation in the cylindrical interpolation mode, cancel any ongoing tool radius/tool nose radius compensation mode before entering the cylindrical interpolation mode. Then, start and terminate tool radius/tool nose radius compensation within the cylindrical interpolation mode.

- Cylindrical interpolation accuracy

In the cylindrical interpolation mode, the amount of travel of a rotary axis specified by an angle is once internally converted to a distance of a linear axis on the outer surface so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, such a distance is converted back to an angle. For this conversion, the amount of travel is rounded to a least input increment.
So when the radius of a cylinder is small, the actual amount of travel can differ from a specified amount of travel. Note, however, that such an error is not accumulative.
If manual operation is performed in the cylindrical interpolation mode with manual absolute on, an error can occur for the reason described above.

$$\text{The actual amount of travel} = \left[\frac{\text{MOTION REV}}{2\times 2\pi\text{R}}\times\left[\text{Specified value}\times\frac{2\times 2\pi\text{R}}{\text{MOTION REV}}\right]\right]$$

MOTION REV : The amount of travel per rotation of the rotary axis (360°)
R : Workpiece radius
[ ] : Rounded to the least input increment

Limitation

- Arc radius specification in the circular interpolation

In the cylindrical interpolation mode, an arc radius cannot be specified with word address I, J, or K.

- Positioning

In the cylindrical interpolation mode, positioning operations (including those that produce rapid traverse cycles such as G28, G53, G73, G74, G76, G80 to G89) cannot be specified. Before positioning can be specified, the cylindrical interpolation mode must be cancelled. Cylindrical interpolation (G07.1) cannot be performed in the positioning mode (G00).

- Cylindrical interpolation mode setting

In the cylindrical interpolation mode, the cylindrical interpolation mode cannot be reset. The cylindrical interpolation mode must be cancelled before the cylindrical interpolation mode can be reset.

- Rotary axis

Only one rotary axis can be set for cylindrical interpolation. Therefore, it is impossible to specify more than one rotary axis in the G07.1 command.

- Rotary axis roll-over

If a rotary axis using the roll-over function is specified at the start of the cylindrical interpolation mode, the roll-over function is automatically disabled in the cylindrical interpolation mode. After the cylindrical interpolation mode is canceled, the roll-over function is enabled automatically.

- Rotary axis control function

If a rotary axis using the multiple rotary axis control function is specified at the start of the cylindrical interpolation mode, the rotary axis control function is automatically disabled in the cylindrical interpolation mode. After the cylindrical interpolation mode is canceled, the rotary axis control function is enabled automatically.

- Tool radius/tool nose radius compensation

If the cylindrical interpolation mode is specified when tool radius/tool nose radius compensation is already being applied, correct compensation is not performed. Specify compensation in the cylindrical interpolation mode.

- Canned cycle for drilling

Canned cycles (G73, G74, and G81 to G89 for M series / G80 to G89 for T series) for drilling, cannot be specified during cylindrical interpolation mode.

M

- Coordinate system setting

In the cylindrical interpolation mode, a workpiece coordinate system (G92, G54 to G59) or local coordinate system (G52) cannot be specified.

- Tool offset

A tool offset must be specified before the cylindrical interpolation mode is set. No offset can be changed in the cylindrical interpolation mode.

- Index table indexing function

Cylindrical interpolation cannot be specified when the index table indexing function is being used.

- Parallel axis

The rotary axis specified for cylindrical interpolation must not be a parallel axis.

T

- Coordinate system setting

In the cylindrical interpolation mode, a workpiece coordinate system G50 cannot be specified.

- Mirror image for double turret

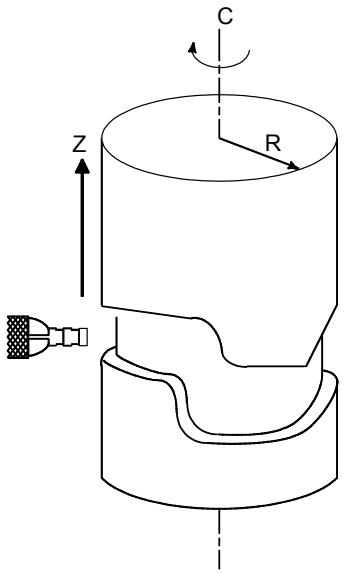
Mirror image for double turret, G68 and G69, cannot be specified during cylindrical interpolation mode.

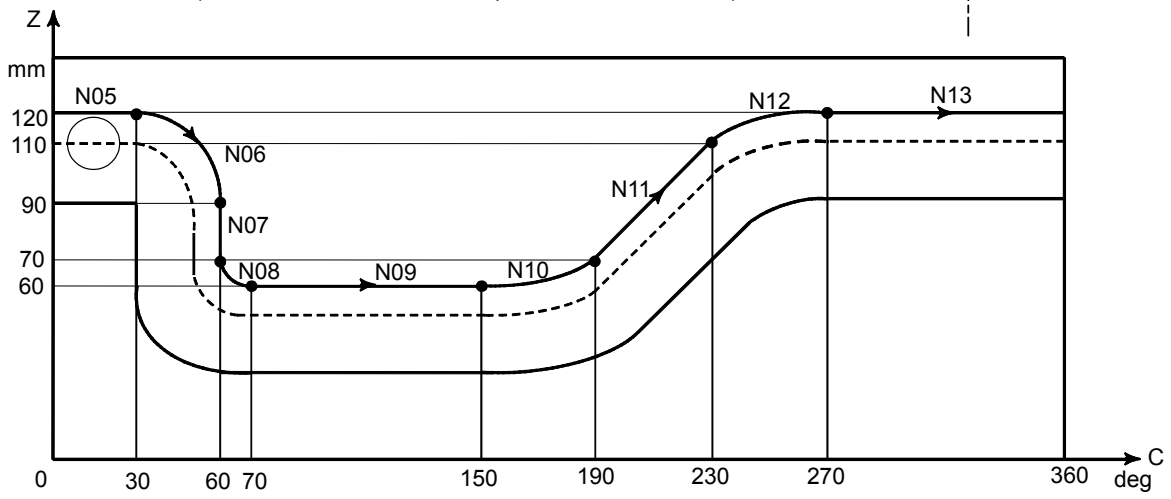
Example

Example of a Cylindrical Interpolation

```
O0001 (CYLINDRICAL INTERPOLATION );
N01 G00 G90 Z100.0 C0 ;
N02 G01 G91 G18 Z0 C0 ;
N03 G07.1 C57299 ;*
N04 G90 G01 G42 Z120.0 D01 F250. ;
N05 C30.0 ;
N06 G03 Z90.0 C60.0 R30.0 ;
N07 G01 Z70.0 ;
N08 G02 Z60.0 C70.0 R10.0 ;
N09 G01 C150.0 ;
N10 G02 Z70.0 C190.0 R75.0 ;
N11 G01 Z110.0 C230.0 ;
N12 G03 Z120.0 C270.0 R75.0 ;
N13 G01 C360.0 ;
N14 G40 Z100.0 ;
N15 G07.1 C0 ;
N16 M30 ;
```

(* A command with a decimal point can also be used.)





4.9.2 Cylindrical Interpolation by Plane Distance Command

Overview

In the conventional rotary axis command in cylindrical interpolation, the angle of the rotary axis is specified.

This function enables the rotary axis command in cylindrical interpolation to be specified by distance on the developed plane by setting parameters.

Format

G07.1 IP r; Starts the cylindrical interpolation mode (enables cylindrical interpolation).

:

G07.1 IP 0; The cylindrical interpolation mode is cancelled.

IP : An address for the rotary axis

r : The radius of the workpiece

Specify G07.1 IPr; and G07.1 IP0; in separate blocks.

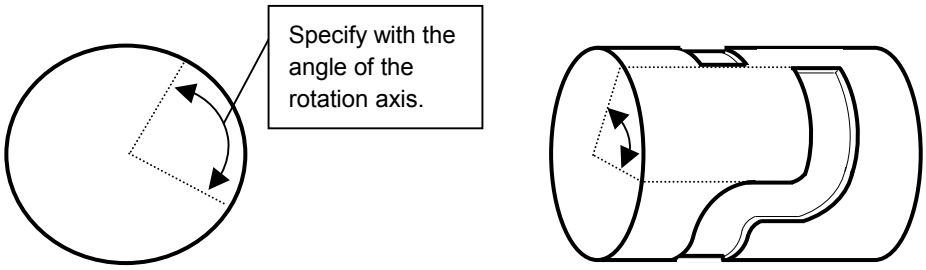
G107 can be used instead of G07.1.

NOTE
Only a positive value is effective as the radius of the workpiece. If a negative value is specified, alarm PS0175 is issued.

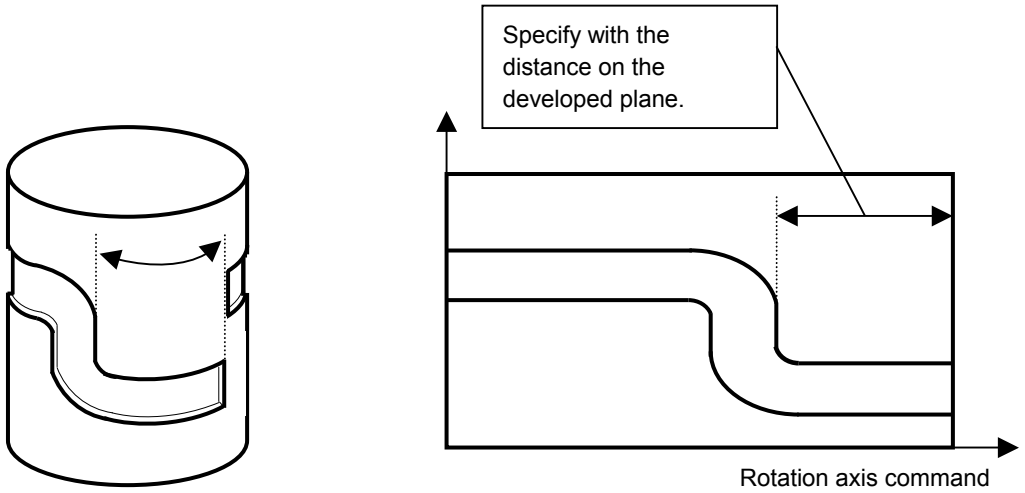
Explanation

By using bit 2 (DTO) of parameter No. 3454, it is possible to switch the rotation axis command during cylindrical interpolation between the angle of the rotation axis and the distance on the developed plane.

- In the case of the angle of the rotation axis (When bit 2 (DTO) of parameter No. 3454 is set to 0)
  The rotation axis command in cylindrical interpolation mode is executed with the angle of the rotation axis. From the program, specify the angle of the rotation axis that corresponds to the specified point on the developed plane.
  The rotation axis command uses the angle of the rotation axis [deg].



- In the case of the distance on the developed plane (When bit 2 (DTO) of parameter No. 3454 is set to 1)
  The rotation axis command in cylindrical interpolation is executed with the distance on the developed plane. The rotation axis command uses the distance on the developed plane and, therefore, the command unit varies depending on which of inch or metric input to use.



Note

NOTE
1 For details of the operation of cylindrical interpolation, as well as limitations, see Subsection, "Cylindrical Interpolation".
2 This function is an optional function.

4.10 CUTTING POINT INTERPOLATION FOR CYLINDRICAL INTERPOLATION (G07.1)

The conventional cylindrical interpolation function controls the tool center so that the tool axis always moves along a specified path on the cylindrical surface, towards the rotary axis (cylindrical axis) of the workpiece. The cutting point interpolation for cylindrical interpolation function controls the tool so that the tangent of the tool and the cutting surface of a contour always passes the rotation center of the workpiece. This means that the cutting surface of the contour is always perpendicular to the cylinder. With this function, the figure on the cutting surface can always be kept constant regardless of the tool radius compensation value of the tool used.

Format

This command is specified in the same way as for the conventional cylindrical interpolation function.

G07.1 IPr ; Circular interpolation mode on start (enabling cylindrical interpolation)

:

G07.1 IP0 ; Circular interpolation mode cancel

IP : One rotary axis address
r : Cylinder radius of rotary axis
Specify each of G07.1 IPr; and G07.1 IP0; singly in a block.
G107 can also be specified.

Explanation

- Comparison with conventional circular interpolation

As shown in Fig. 4.10 (a), the tool is controlled in the offset axis (Y-axis) direction that is perpendicular to the tool center and workpiece rotation center.

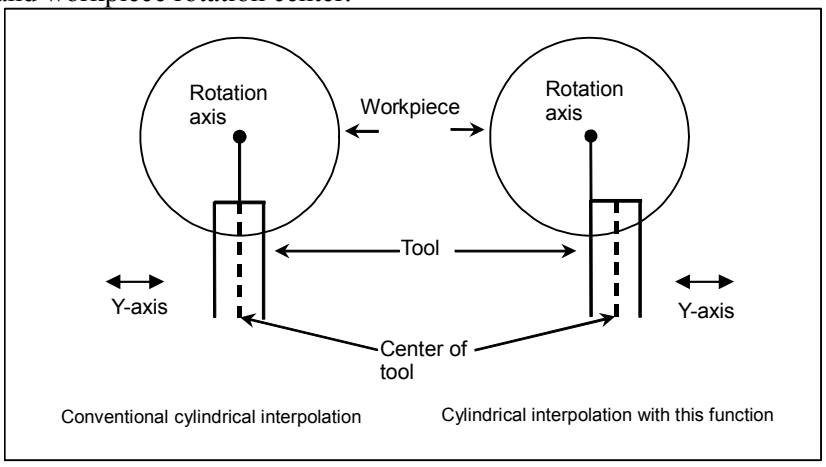


Fig. 4.10 (a) Comparison with conventional circular interpolation

- Cutting point compensation

(1) Cutting point compensation between blocks

As shown in Fig. 4.10 (b), cutting point compensation is achieved by moving between blocks N1 and N2.

(a) Let C1 and C2 be the heads of the vectors normal to N1 and N2 from S1, which is the intersection of the tool center paths of blocks N1 and N2

(b) After the tool moves to S1 according to the command of N1, the tool moves through V on the C-axis as a result of cutting point compensation, then through $-V \times \frac{\pi}{180} \times r$ along the Y-axis.

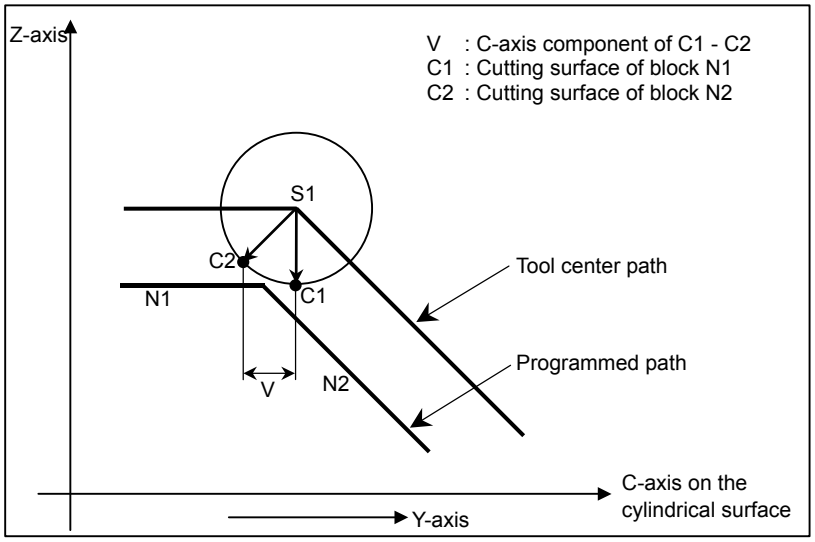


Fig. 4.10 (b) Cutting point compensation between blocks

(2) Cutting point compensation in a circular command block

As shown in Fig.4.10 (c), the movement required for cutting point compensation is made simultaneously with circular interpolation in block N1.

(a) Let C0 be the head of the vector normal to N1 from S0, which is the tool center position at the start point of circular block N1. Let C1 be the head of the similar vector at the end point.

(b) As the tool moves from S0 to S1, a superimposed movement is made by the C-axis component of (C1 - C2) (V in the figure) on the C-axis, and a superimposed movement is made by $-\mathrm{V}\times\frac{\pi}{180}\times\mathrm{r}$ along the Y-axis. along the Y-axis.

That is, the following expressions are valid. As movement is made through L as shown in Fig.4.10 (c), the superimposed movements are made on the C-axis and Y-axis as follows:

$$\Delta C = \Delta V$$

$$\Delta Y = -\frac{\pi}{180}(\Delta V)r$$

ΔV : Cutting point compensation value (ΔV2 - ΔV1) for movement of ΔL

ΔV1 : C-axis component of the vector normal to N1 from the tool center of the start point of ΔL

ΔV2 : C-axis component of the vector normal to N1 from the tool center of the end point of ΔL

R : Arc radius

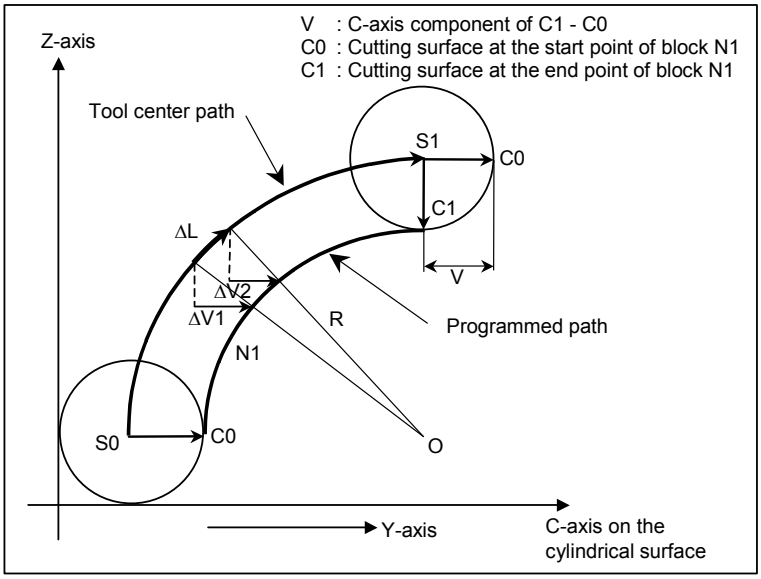


Fig.4.10 (c) Cutting point compensation in a circular command block

(3) When cutting point compensation is not applied between blocks
When, as shown in Fig.4.10 (d) and Fig.4.10 (e), the cutting point compensation value (V in the Fig.4.10 (d) and Fig.4.10 (e)) is less than the value set in parameter No. 19534, one of the operations below is performed. (The operation that is performed depends on the setting of bit 6 (CYS) of parameter No.19530.

(a) When bit 6 (CYS) of parameter No.19530 is set to 1
Cutting point compensation is not applied between blocks N1 and N2, but is applied when block N2 is executed.

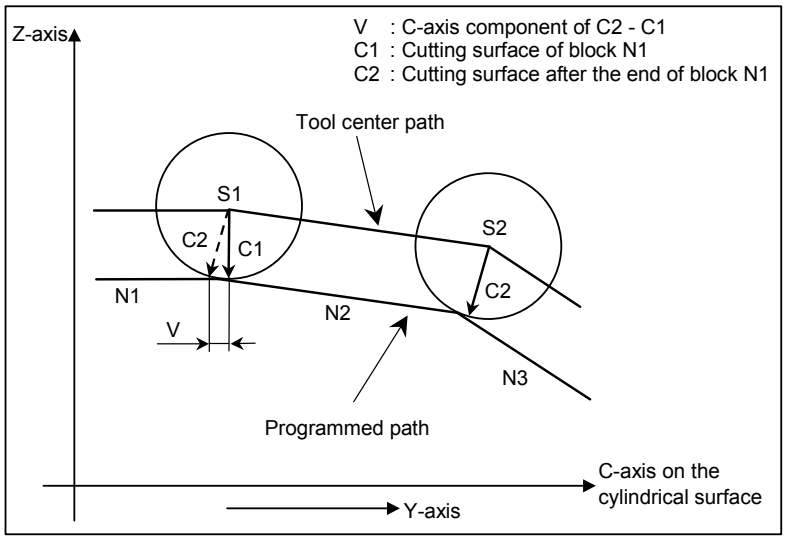


Fig.4.10 (d) When bit 6 (CYS) of parameter No.19530 is set to 1

(b) When bit 6 (CYS) of parameter No.19530 is set to 0
Cutting point compensation is not performed between blocks N1 and N2. Whether to apply cutting point compensation between block N2 and N3 is determined by taking the cutting point compensation value between blocks N2 and N3 (V in the Fig.4.10 (e)) into consideration.

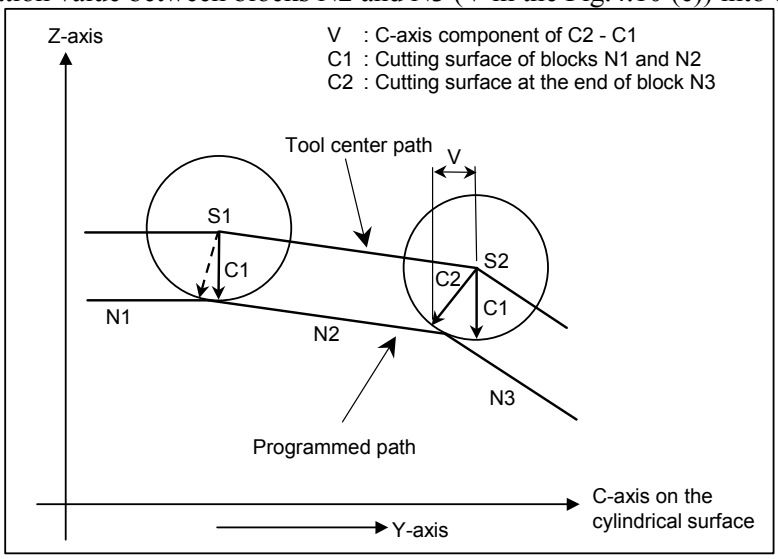


Fig.4.10 (e) When bit 6 (CYS) of parameter No.19530 is 0

(c) When the amount of travel (L1) of block N2 is less than the value set in parameter No. 19535, as shown in Fig.4.10 (f), cutting point compensation is not applied between blocks N1 and N2. Instead, block N2 is executed with the cutting point compensation of the previous block. When the amount of travel (L2) of block N3 is greater than the value set in parameter No. 19535, cutting point compensation is applied between blocks N2 and N3.

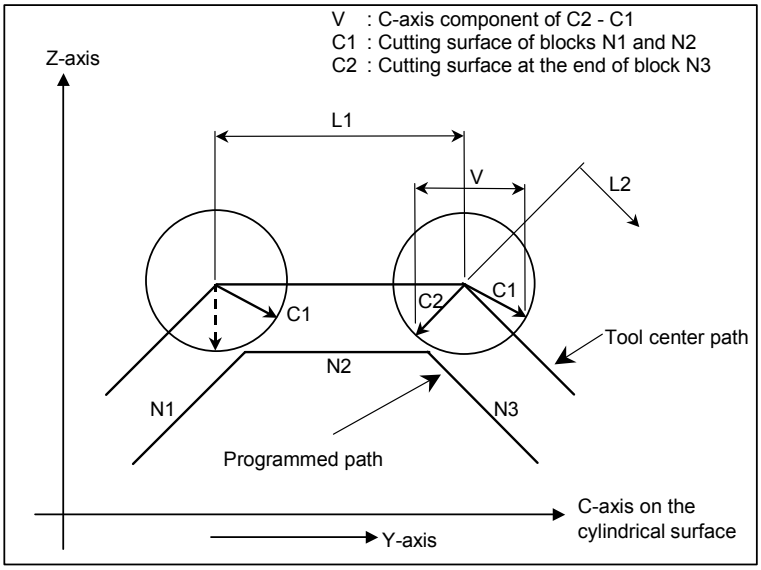


Fig.4.10 (f) When the amount of travel (L1) of block N2 is less than the parameter value

(d) When, as shown in Fig.4.10 (g), the diameter of an arc (R in the figure) is less than the value set in parameter No. 19535, cutting point compensation is not applied simultaneously with circular interpolation

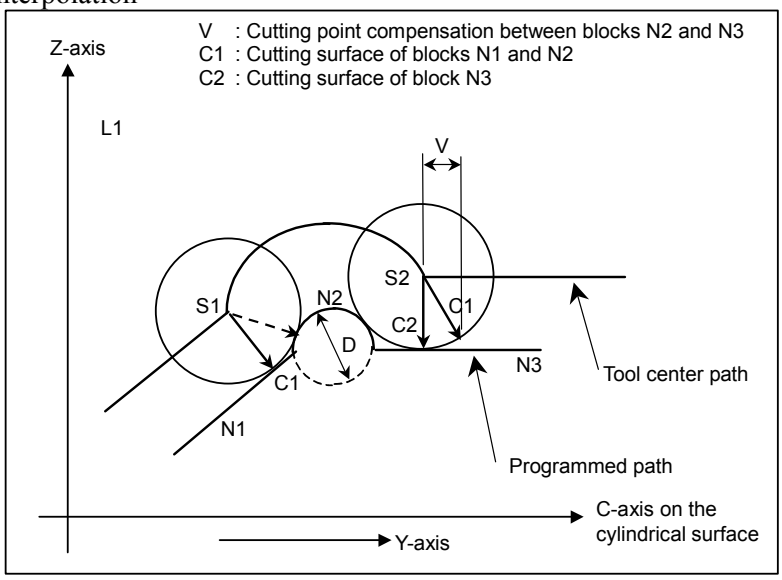


Fig.4.10 (g) When the diameter of an arc is less than the parameter value

- When cutting point compensation is used with normal direction control

When cutting point compensation is used with normal direction control, cutting point compensation between specified blocks is performed regardless of the method described in "**Cutting point compensation**" above, but is performed simultaneously with the movement of the normal direction control axis (C-axis).

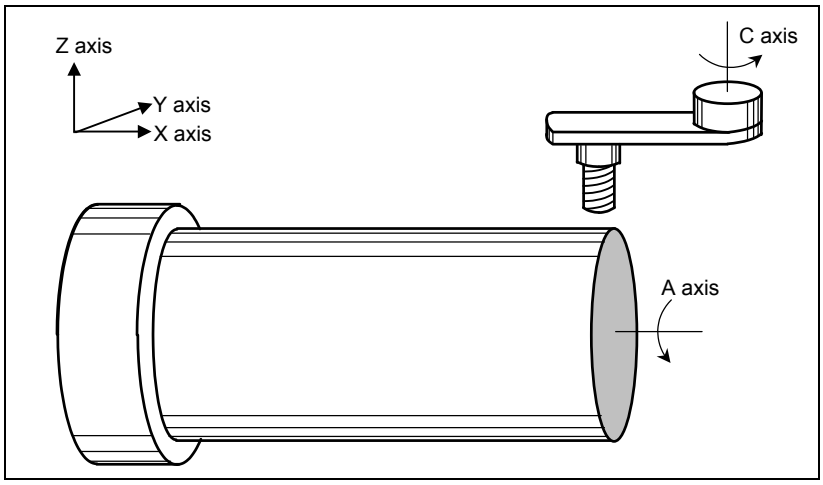


Fig. 4.10 (h) When used with normal direction control

(1) When the normal direction changes between blocks N1 and N2, cutting point compensation is also performed between blocks N1 and N2.
As shown in Fig. 4.10 (i), cutting point compensation described in (1) in "**Cutting point compensation**" is performed simultaneously with the movement made by normal direction control between blocks N1 and N2.

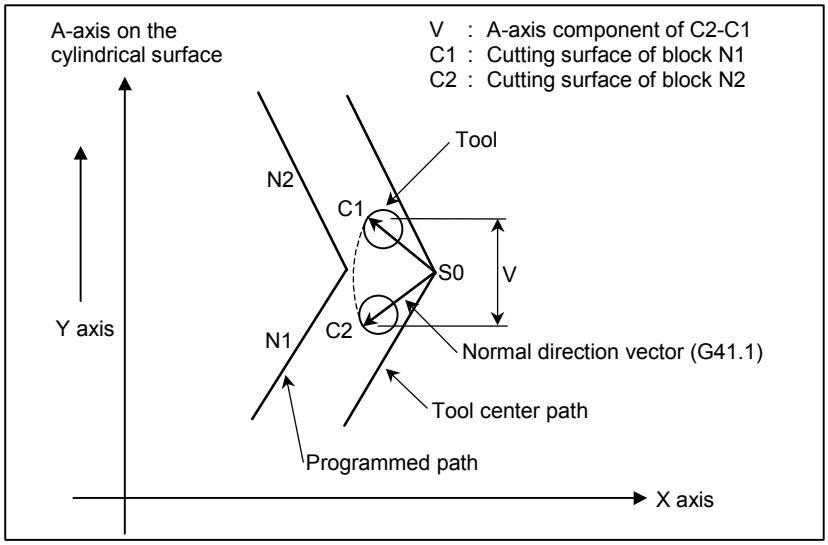


Fig. 4.10 (i) When the normal direction changes between blocks N1 and N2

(2) When gentle normal direction control changes the normal direction as a specified block is executed, cutting point compensation is performed simultaneously with the movement made by the block. When the normal direction control axis rotates by θ1 simultaneously with the movement made by blocks N1 to N2 as shown in Fig. 4.10 (j), cutting point compensation by the movement of vector V1 is also performed simultaneously with the movement in N2.

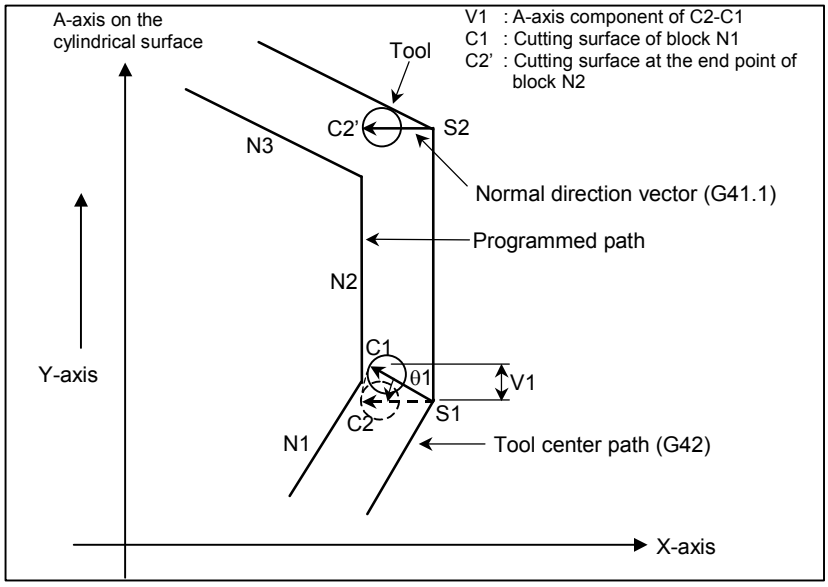


Fig. 4.10 (j) Gentle normal direction control

(3) When a specified block is executed while the normal direction control axis is held in the normal direction set at the end point of the previous block, cutting point compensation is not performed, and cutting point compensation applied in the previous block is maintained. As shown in Fig. 4.10 (k), when the movement amount in N2 (L1 in the figure) is smaller than the setting of parameter No. 5483, so the normal direction control axis does not rotate at S1 while the movement amount in N3 (L2 in the figure) is greater than the setting of parameter No. 5483, so the normal direction axis rotates at S2, cutting point compensation is not performed at S1 but is performed by the movement for vector V2 at S2.

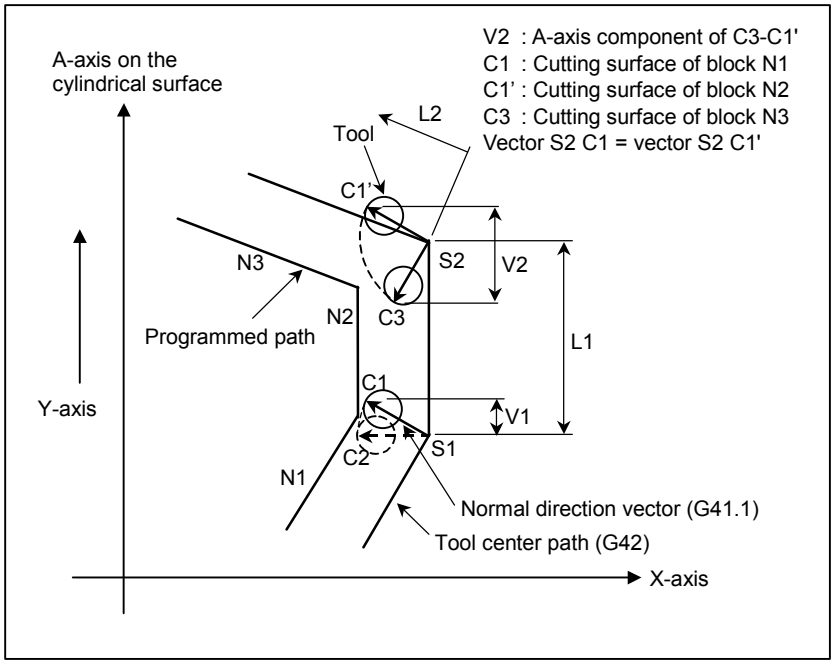


Fig. 4.10 (k) When the normal direction is the same as in the previous block

- Feedrate during cutting point compensation

(1) The tool moves at a specified feedrate while cutting point compensation is being applied between blocks.

(2) The actual speed indication and feedrate during circular interpolation are as described below.
Actual speed indication
The speed component of each axis after cutting point compensation at a point in time during circular interpolation is as follows:

$Fz' = Fz$: Speed component of linear axis

$Fc' = Fc + (Vce - Vcs)$: Speed component of rotary axis

$Fy' = -(Vce - Vcs)\frac{\pi r}{180}$: Speed component of offset axis

Fz : Speed component of a cylindrical interpolation linear axis before cutting point compensation
Fc : Speed component of cylindrical interpolation rotation axis before cutting point compensation
Vcs: Rotation axis component of a tool contact point vector (Vs in the figure) at the start point at a point in time
Vce: Rotation axis component of tool contact point vector (Ve in the figure) at the end point at a point in time
r : Radius of the cylinder of a rotary axis
Accordingly, the actual speed indication during circular interpolation is greater than the specified value when |Fc'| > |Fc|(inner offset of the arc). Conversely, the actual speed indication during circular interpolation is less than the specified value when |Fc'| < |Fc|(outer offset of the arc).

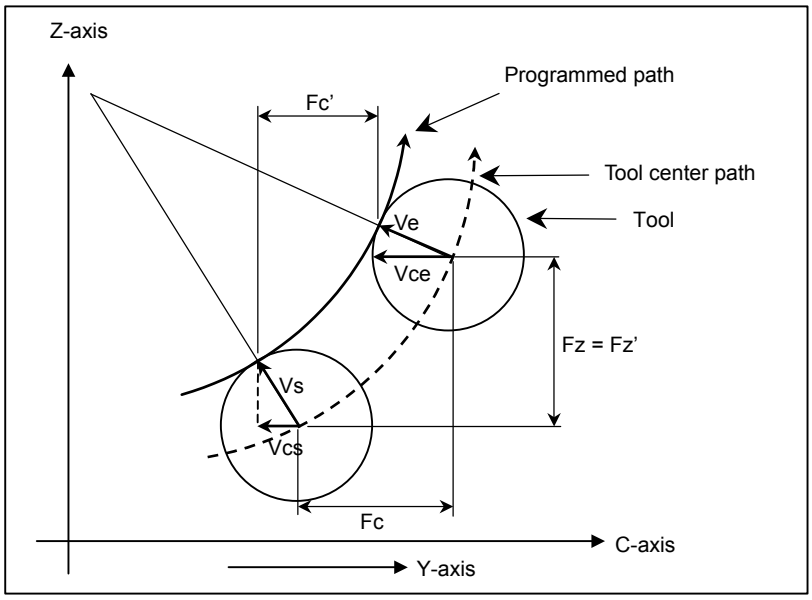


Fig.4.10 (l) Actual speed indication during circular interpolation

- Usable G codes

(1) In any of the following G code modes, cutting point interpolation for cylindrical interpolation can be specified:
G01, G02, G03 : Linear interpolation, circular interpolation
G17, G18, G19 : Plane selection
G22 : Stored stroke check function on
G64 : Cutting mode
G90, G91 : Absolute programming, incremental programming
G94 : Feed per minute

(2) Any of the following G codes can be specified in cutting point interpolation for cylindrical interpolation mode:
G01, G02, G03 : Linear interpolation, circular interpolation
G04 : Dwell
G40, G41, G42 : Tool radius compensation
G40.1 to G42.1 : Normal direction control

G64 : Cutting mode
G65 to G67 : Macro call
G90, G91 : Absolute programming, incremental programming

- Parameter

To enable this function, set bit 5 (CYA) of parameter No. 19530 to 1.

Limitation

- Overcutting during inner corner cutting

Theoretically, when the inner area of a corner is cut using linear interpolation as shown in Fig.4.10 (m), this function slightly overcuts the inner walls of the corner. This overcutting can be avoided by specifying a value of R that is slightly greater than the radius of the tool at the corner.

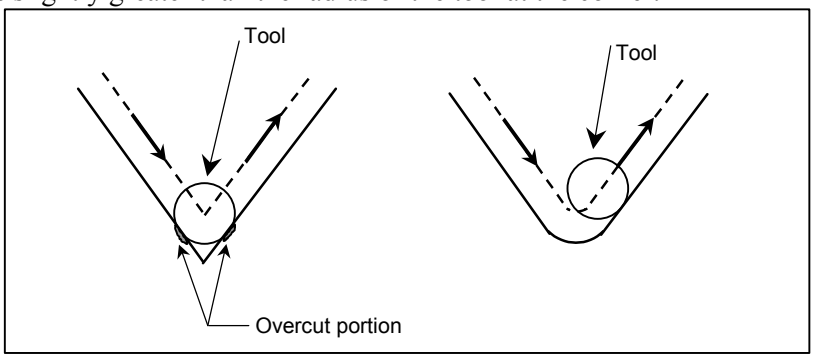


Fig.4.10 (m) Overcutting

- Setting the minimum input increment for an offset axis (Y-axis)

Set the same minimum input increment for an offset axis and linear axis when cylindrical interpolation is performed.

- Workpiece radius specification

When specifying the radius of a workpiece, use the minimum input increment (with no decimal point) for the linear axis used in cylindrical interpolation.

- Reference axis setting (parameter No. 1031)

When different increment systems are used for a linear axis and a rotary axis in cylindrical interpolation, specify the number of the linear axis in cylindrical interpolation to set the axis as the reference axis.

- Rotary axis roll-over function

When a rotary axis for which the rollover function is used is specified as a rotary axis to be used for cylindrical interpolation, the roll-over function is disabled in the cylindrical interpolation mode.
When the cylindrical interpolation is canceled, the rollover function is automatically enabled.

- Program restart

In a program restart operation, G07.1 for the cylindrical interpolation mode must not be specified. Otherwise, PS0175 is issued.

Example

- Example of cutting point interpolation for cylindrical interpolation

The sample program below indicates the positional relationships between a workpiece and tool.

```
O0001 (CYLINDRICAL INTERPOLATION1) ;
N01 G00 G90 Z100.0 C0 ;
N02 G01 G91 G19 Z0 C0 ;
N03 G07.1 C57299 ;
N04 G01 G42 G90 Z120.0 D01 F250. ;.....(1)
```

```
N05 C20.0 ;  .....(2)
N06 G02 Z110.0 C60.0 R10.0 ; .....(3)
N07 G01 Z100.0 ;.....(4)
N08 G03 Z60.0 C70.0 R40.0 ; .....(5)
N09 G01 C100.0 ;
   :
M30 ;
```

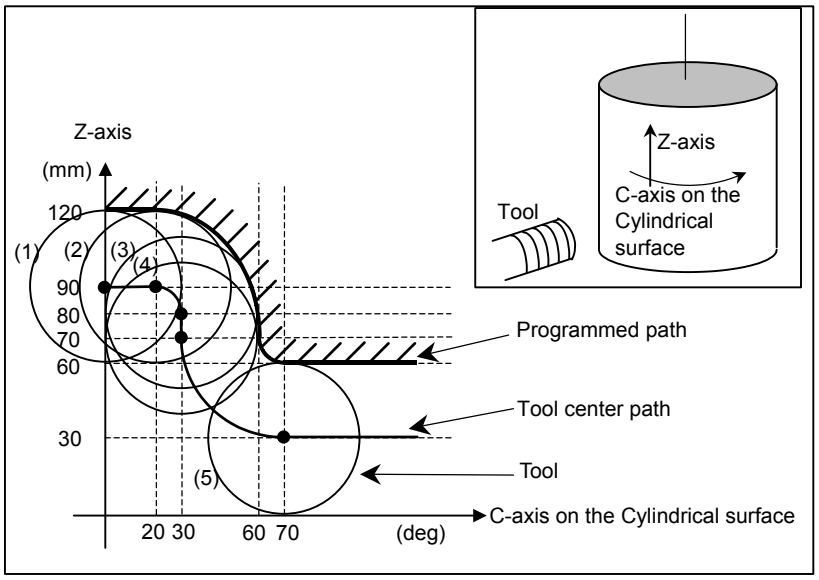


Fig. 4.10 (n) Path of sample program for cutting point interpolation for cylindrical interpolation

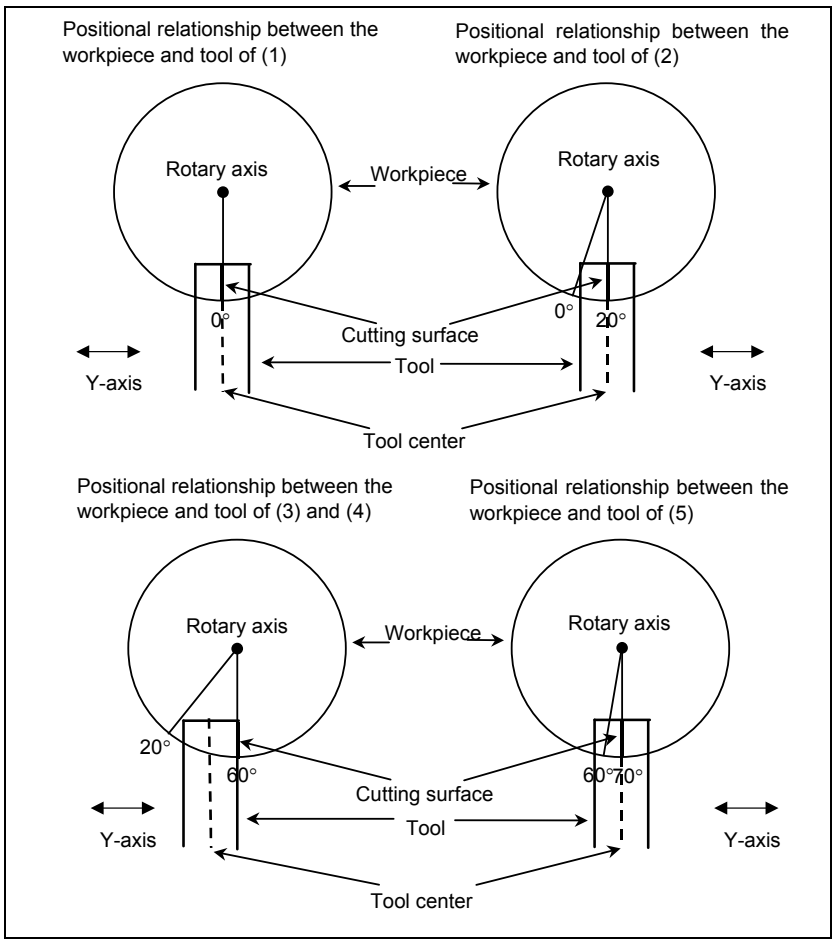


Fig. 4.10 (o) Positional relationships between workpiece and tool of sample program

The cutting surface in the rotary axis direction in (3) and (4) are uniform even if the tool radius compensation amount is modified.

- Example of specifying cutting point interpolation for cylindrical interpolation and normal direction control at the same time

Tool radius compensation No.01 is 30 mm.

```
O0002 (CYLINDRICAL INTERPOLATION2) ;
N01 G00 G90 X100.0 A0 ;
N02 G01 G91 G17 X0 A0 ;
N03 G07.1 C57299 ;
N04 G01 G41 G42.1 G90 X120.0 D01 F250. ;
N05 A20.0 ;
N06 G03 X80.0 A60.0 R40.0 ;
N07 G01 X70.0 ;
N08 G02 X70.0 A70.0 R10.0 ;
N09 G01 A150.0 ;
N10 G02 X70.0 A190.0 R85.0 ;
N11 G01 X110.0 A265.0 ;
N12 G03 X120.0 A305.0 R85.0 ;
N13 G01 A360.0 ;
N14 G40 G40.1 X100.0 ;
N15 G07.1 A0 ;
N16 M30 ;
```

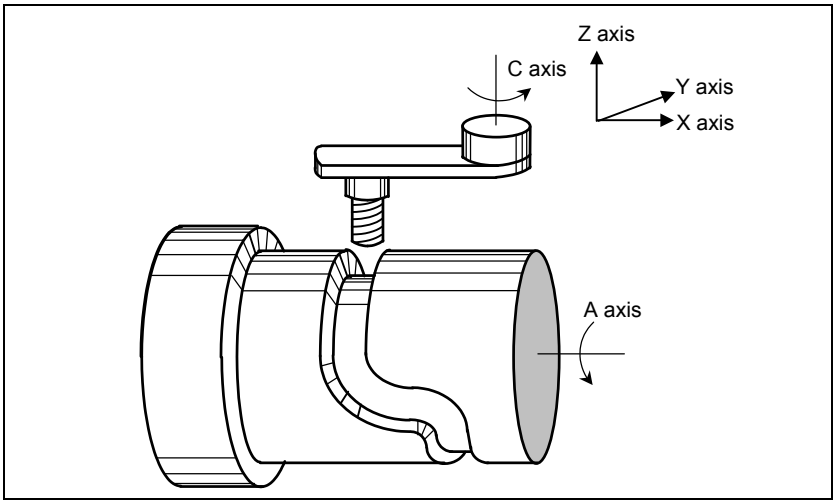


Fig. 4.10 (p) Example of specifying normal direction control at the same time

4.11 EXPONENTIAL INTERPOLATION (G02.3, G03.3)

Exponential interpolation exponentially changes the rotation of a workpiece with respect to movement on the rotary axis. Furthermore, exponential interpolation performs linear interpolation with respect to another axis. This enables tapered groove machining with a constant helix angle (constant helix taper machining). This function is best suited for grooving and grinding tools such as taper end mills.

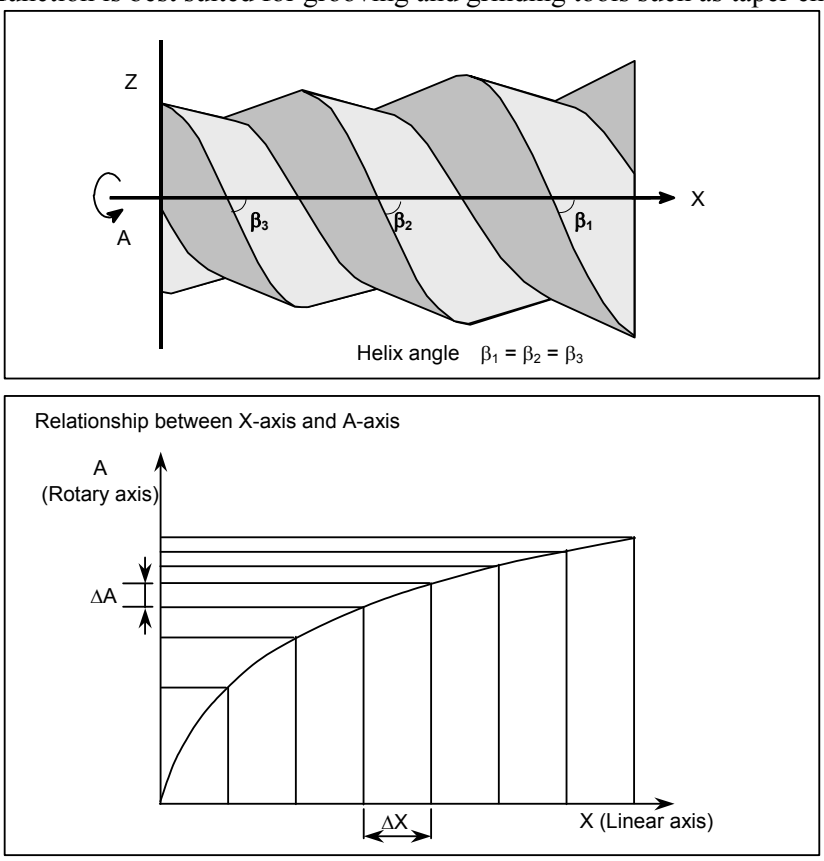


Fig. 4.11 (a) Exponential interpolation

Format

Positive rotation (ω = 0)

G02. 3 X_ Y_ Z_ I_ J_ K_ R_ F_ Q_ ;

Negative rotation (ω = 1)

G03. 3 X_ Y_ Z_ I_ J_ K_ R_ F_ Q_ ;

X_ : Specifies an end point with an absolute or incremental value.
Y_ : Specifies an end point with an absolute or incremental value.
Z_ : Specifies an end point with an absolute or incremental value.
I_ : Specifies angle I (from ±1 to ±89 deg in units of 0.001 deg).
J_ : Specifies angle J (from ±1 to ±89 deg in units of 0.001 deg).
K_ : Specifies the amount to divide the linear axis for exponential interpolation (span value). The specification unit depends on the reference axis. Specify a positive value. The span value is specified in the manner specified in bit 0 (SPN) of parameter No. 5630. If SPN is set to 0, the division amount is specified in parameter No. 5643. If SPN is set to 1, the value specified with K becomes valid.
R_ : Specifies constant R for exponential interpolation. (See Explanations below.)
F_ : Specifies the initial feedrate.
Specified in the same way as an ordinary F code. Specify a composite feedrate including a feedrate on the rotary axis.
Q_ : Specifies the feedrate at the end point.
The same unit used for F is used. The CNC internally performs interpolation between the initial feedrate (F) and final feedrate (Q), depending on the travel distance on the linear axis.

Explanation

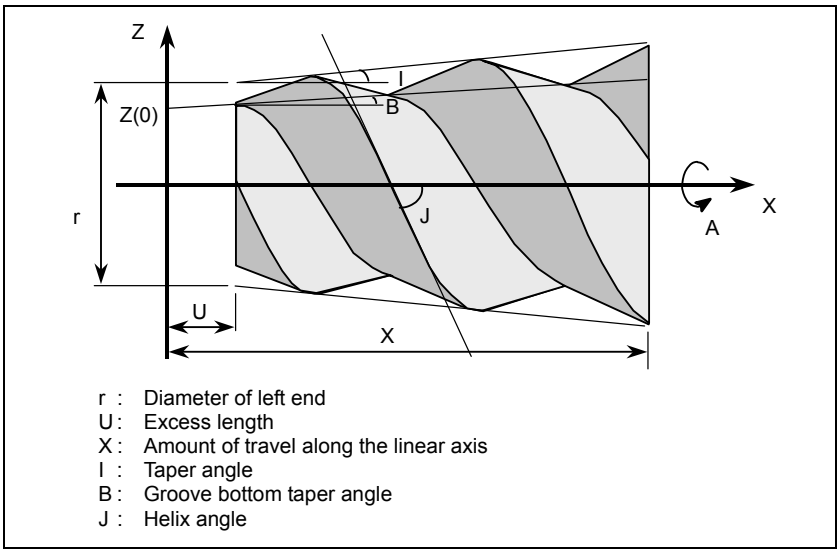


Fig. 4.11 (b) Constant helix machining for producing a tapered figure

In Fig. 4.11 (b) an absolute value on the X-axis, Z-axis, or A-axis is expressed as a function of workpiece rotation angle θ, such as $X(\theta)$, $Z(\theta)$, and $A(\theta)$.
Linear interpolation with the X-axis is performed for an axis other than the X-axis or A-axis. When $X(\theta) = 0$, $A(\theta) = 0$.

The relationship is expressed as follows.

$$Z(\theta) = \left\{ \frac{r}{2} - U * \tan(I) \right\} * (e^{\frac{\theta}{K}} - 1) * \frac{\tan(B)}{\tan(I)} + Z(0) \quad \cdots\cdots\cdots\cdots (1)$$

$$X(\theta) = \left\{ \frac{r}{2} - U * \tan(I) \right\} * (e^{\frac{\theta}{K}} - 1) * \frac{1}{\tan(I)} \quad \cdots\cdots\cdots\cdots (2)$$

$$A(\theta) = (-1)^{\omega} * \frac{360}{2\pi} * \theta$$

Where

$$K = \frac{\tan J}{\tan I}$$

ω: Helix direction (0: Positive, 1: Negative)

From Expressions (1) and (2), the following is obtained ;

$$Z(\theta) = \tan(B) * X(\theta) + Z(0) \quad \cdots\cdots\cdots\cdots (3)$$

From Expression (3), the Z-axis position is determined from a groove bottom taper angle (B) and X-axis position.

From Expression (1) and exponential definition expression (described later), the following is determined:

$$R = r/2 - U * \tan(I) \quad \cdots\cdots\cdots\cdots (4)$$

Constant R is determined from the left end diameter (r), excess length (U), and taper angle (I) according to Expression (4). Specify a taper angle (I) in address I, and specify a helix angle (J) in address J. Select a helix direction with G02.3 or G03.3.

- Exponential definition expressions

Exponential relational expressions for a linear axis and rotary axis are defined as follows:

$$X(\theta) = R * (e^{\frac{\theta}{K}} - 1) * \frac{1}{\tan(I)} \quad \cdots\cdots\cdots\cdots (5)$$

$$A(\theta) = (-1)^{\omega} * 360 * \frac{\theta}{2\pi} \quad \cdots\cdots\cdots\cdots (6)$$

Where,

$$K = \frac{\tan(J)}{\tan(I)}$$

$$\omega = 0/1$$

R, I, and J are constants, and θ represents an angle (radian).

- Span value K

A movement on an axis is carried out as linear interpolation in units of values obtained by dividing the movement on the X-axis by the span value (address K).

The following is obtained from Expression (5)

$$\theta(X) = K * \ln(\frac{X * \tan(I)}{R} + 1) \quad \cdots\cdots\cdots\cdots (7)$$

When there is movement from X_1 to X_2 on the linear axis, the amount of movement on the rotary axis is determined by:

$$\Delta\theta = K * \{\ln(\frac{X_2 * \tan(I)}{R} + 1) - \ln(\frac{X_1 * \tan(I)}{R} + 1)\}$$

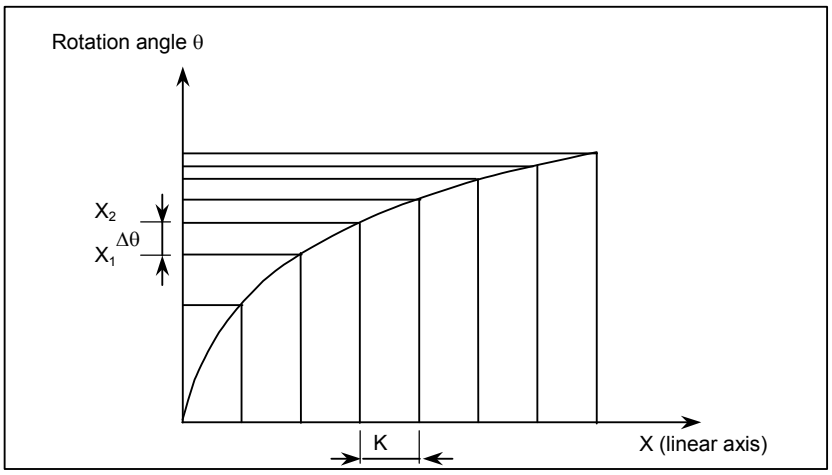


Fig. 4.11 (c) Span value K

- Rotation axis θ

In exponential interpolation, Expression (7) indicates the relationship between the X coordinate and the rotation angle θ about the A-axis. The expression in the parentheses of the natural logarithm ln in Expression (7) must satisfy Expression (8) indicated below, because of the condition of ln (the value in the parentheses is positive).

$$\frac{X * \tan(I)}{R} > -1 \quad \text{.....(8)}$$

If the value of X*tan(I)/R becomes -1 or less, the position comes to the right of point (A) in Fig. 4.11 (d). Because the resultant figure is unfeasible, an alarm PS5062 will be issued.

In exponential interpolation, the X, Y, Z, and U values are handled as coordinates in the workpiece coordinate system. If a positive value specified in incremental programming corresponds to a negative value in the workpiece coordinate system, the negative value is used in the calculation.

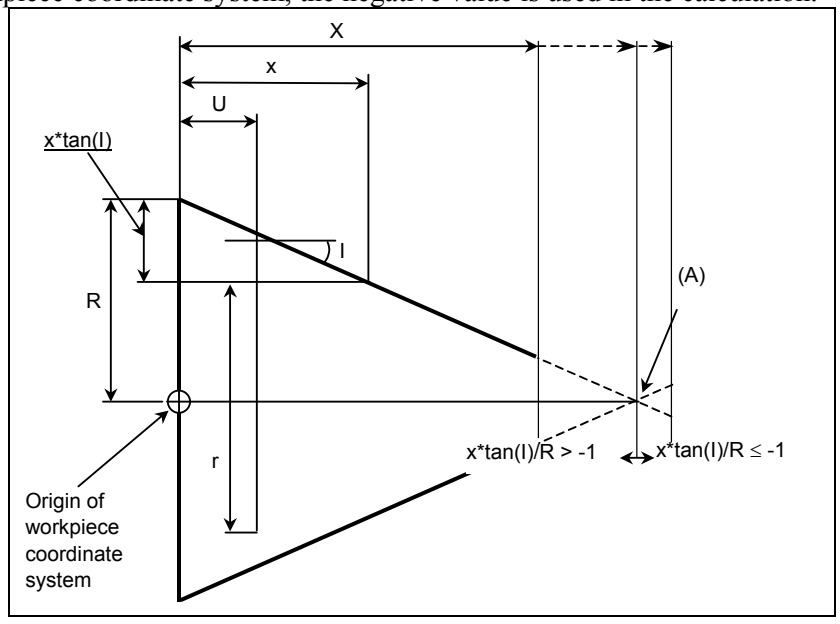


Fig. 4.11 (d) Rotation angle θ

- Taper angle I

The machining profile and the sign of taper angle I have the following relationships:

- If the profile tapers up toward the right, the I value is positive.
- If the profile tapers down toward the right, the I value is negative.

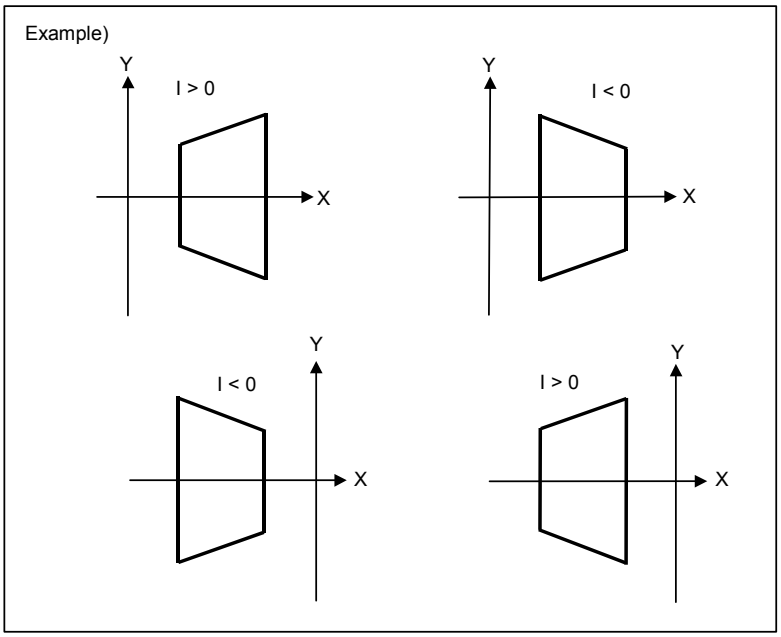


Fig. 4.11 (e) Taper angle I

- Helix angle J

The sign of the helix angle J is assigned as illustrated below.

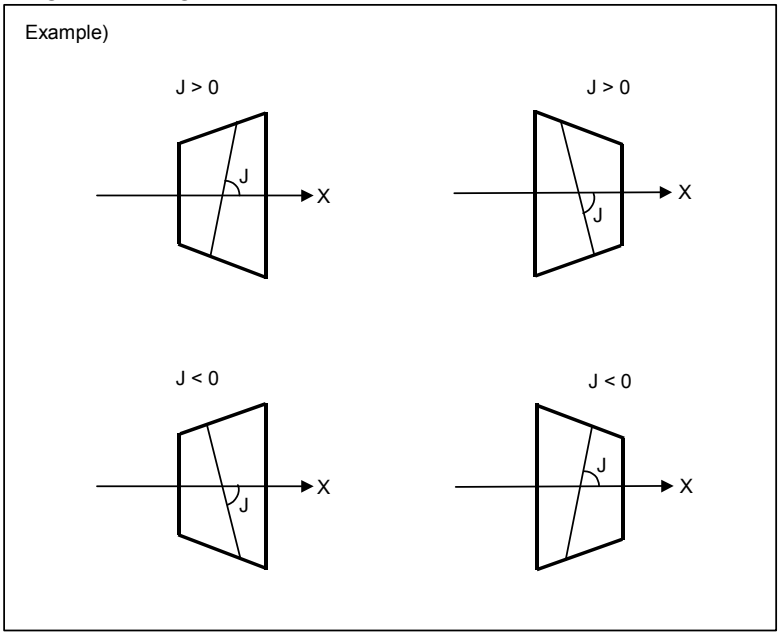
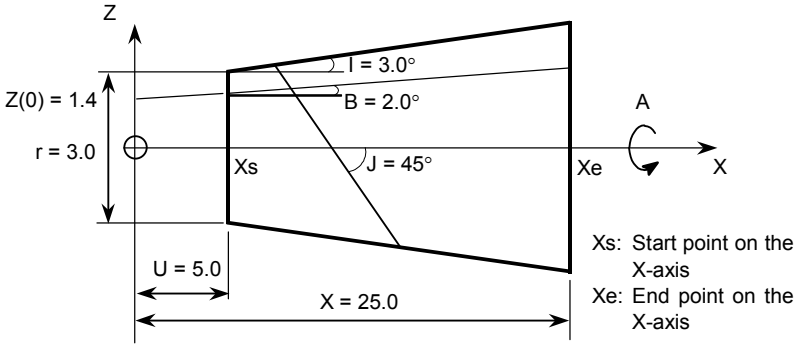


Fig. 4.11 (f) Helix angle J

Example




```
N10 G90 G01 X5.0 Z1.575 ;
N20 G02.3 X25.0 Z2.273 I3.0 J-45.0 K1.0 R1.238
F1000 Q1000 ;
```

The start point and end point of the Z-axis and the constant R are obtained from the following expressions:

Start point on the Z-axis $= \tan(B) * Xs + Z(0)$

End point on the Z-axis $= \tan(B) * Xe + Z(0)$

$R = r/2 - U * \tan(I)$

Limitation

- Cases where linear interpolation is performed

Even when the G02.3 or G03.3 mode is set, linear interpolation is performed in the following cases:

- When the linear axis specified in parameter No. 5641 is not specified, or the amount of movement on the linear axis is 0
- When the rotary axis specified in parameter No. 5642 is specified
- When the amount for dividing the linear axis (span value) is 0

- Tool compensation

The tool compensation functions (tool length compensation, tool radius/tool nose radius compensation, and 3-dimensional cutter compensation) cannot be used in the G02.3 or G03.3 mode.

> ⚠ **CAUTION**
> The amount for dividing the linear axis for exponential interpolation (span value) affects figure precision. However, if an excessively small value is set, the machine may stop during interpolation. Try to specify an optimal span value depending on the machine being used.

4.12 SMOOTH INTERPOLATION (G05.1)

Either of two types of machining can be selected, depending on the program command.

- For those portions where the accuracy of the figure is critical, such as at corners, machining is performed exactly as specified by the program command.
- For those portions having a large radius of curvature where a smooth figure must be created, points along the machining path are interpolated with a smooth curve, calculated from the polygonal lines specified with the program command (smooth interpolation).

In this manner, high-speed, high-precision machining can be performed.
For this function, the AI contour control option is required.

Format

G05.1Q2 Xp0 Yp0 Zp0 ; Smooth interpolation mode on
:
G05.1Q0 ; Smooth interpolation mode off

Xp : X axis or an axis parallel to the X-axis
Yp : Y axis or an axis parallel to the X-axis
Zp : Z axis or an axis parallel to the X-axis

NOTE
1 Specify G05.1 alone in a block.
(Avoid specifying any other G code in the same block.)
2 Specify position 0 for the axis programmed in the smooth interpolation mode on block. The specified axis is subjected to smooth interpolation, but no movement is made even in the absolute programming mode.
(Axis moving is not performed in the G05.1Q2 block.)
3 Smooth interpolation mode is also turned off at a reset.

In the G5.1 Q2 block, specify the axis subject to smooth interpolation. Note that up to three axes can be subject to the smooth interpolation command at a time and that only the following axes can be specified.

- Basic three axes (X,Y,Z)
- Axes parallel to the basic three axes

If specifying the machining condition selecting function, specify G5.1 Q1 Rx first and then smooth interpolation.

[Example]

```
O0010                          If the following functions are required
…                              before smooth interpolation, specify G5.1.
(G5.1 Q1 R1;) ———— - AI contour control
                    - Machining condition selecting function
G5.1 Q2 X0 Y0 Z0;
…
G5.1 Q0; ———— Smooth interpolation mode off
              AI contour control mode off
…
M30;
```

- Using smooth interpolation together with tool length compensation

If using smooth interpolation together with tool length compensation (G43, G44) first, and then enable smooth interpolation (G5.1 Q2).

To cancel them, cancel smooth interpolation (G5.1 Q0) first, and then cancel tool length compensation (G49).

[Example]

```
O0010                    If the following functions are required
…                        before smooth interpolation, specify G5.1.
(G5.1 Q1 R1;)            - AI contour control
                         - Machining condition selecting function
G43 H1;
G5.1 Q2 X0 Y0 Z0;
…
G5.1 Q0;
G49;                                  Tool length compensation
…
M30;                     Smooth interpolation
```

Smooth interpolation cannot be used together with tool center point control. If an attempt is made to use them together, alarm PS5421 is issued.

Explanation

- Characteristics of smooth interpolation

To machine a part having sculptured surfaces, such as metal moldings used in automobiles and airplanes, a part program usually approximates the sculptured surfaces with minute line segments. As shown in the following figure (Fig. 4.12 (a)), a sculptured curve is normally approximated using line segments with a tolerance of about 10 μm.

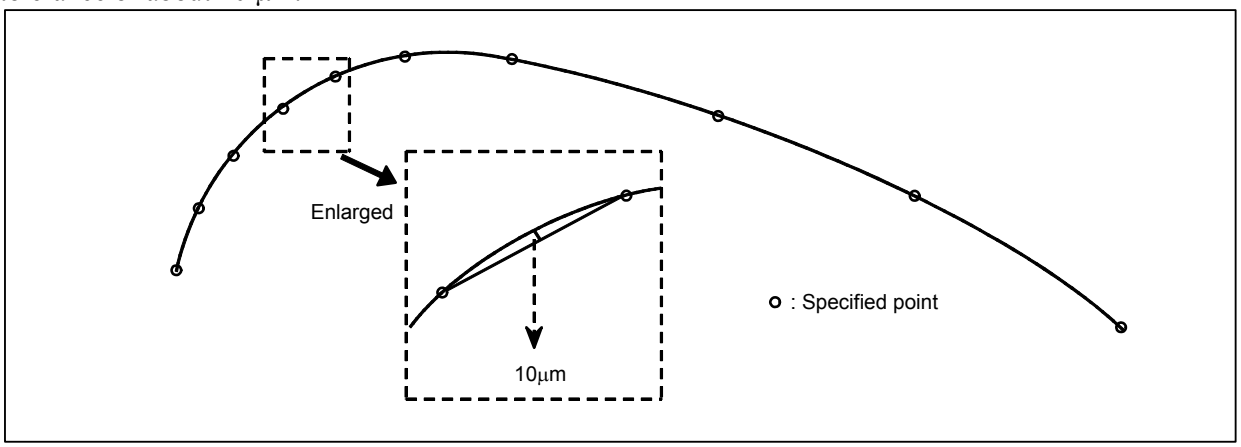
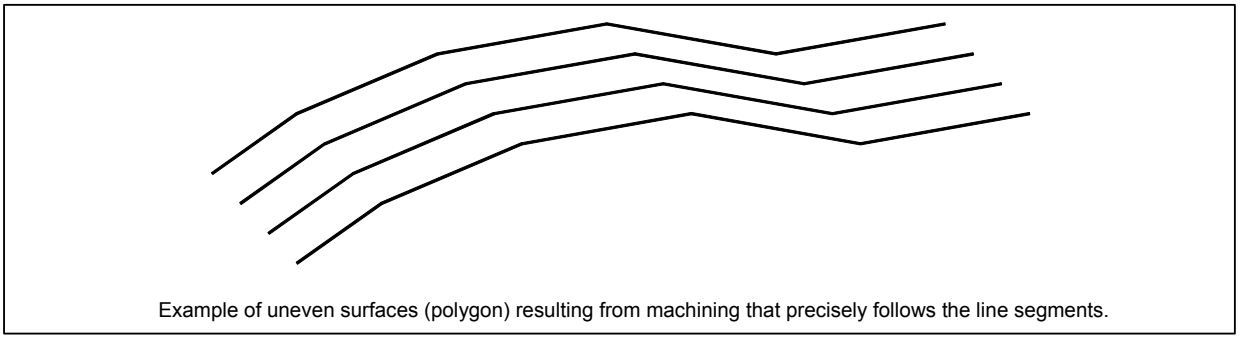


Fig. 4.12 (a)

When a program approximates a sculptured curve with line segments, the length of each segment differs between those portions that have mainly a small radius of curvature and those that have mainly a large radius of curvature. The length of the line segments is short in those portions having a small radius of curvature, while it is long in those portions having a large radius of curvature. Because linear interpolation controls the tool movement exactly as programmed so as not to let the tool depart from the path specified by a part program, machining is performed exactly along line segments with which a sculptured curve is approximated. Consequently, when a curve with a large radius of curvature and a moderate curvature change is machined, the corners of line segments may become apparent. Such concave and convex portions, produced as a result of machining performed exactly as specified, are found troublesome when a smooth surface must be obtained by finishing.

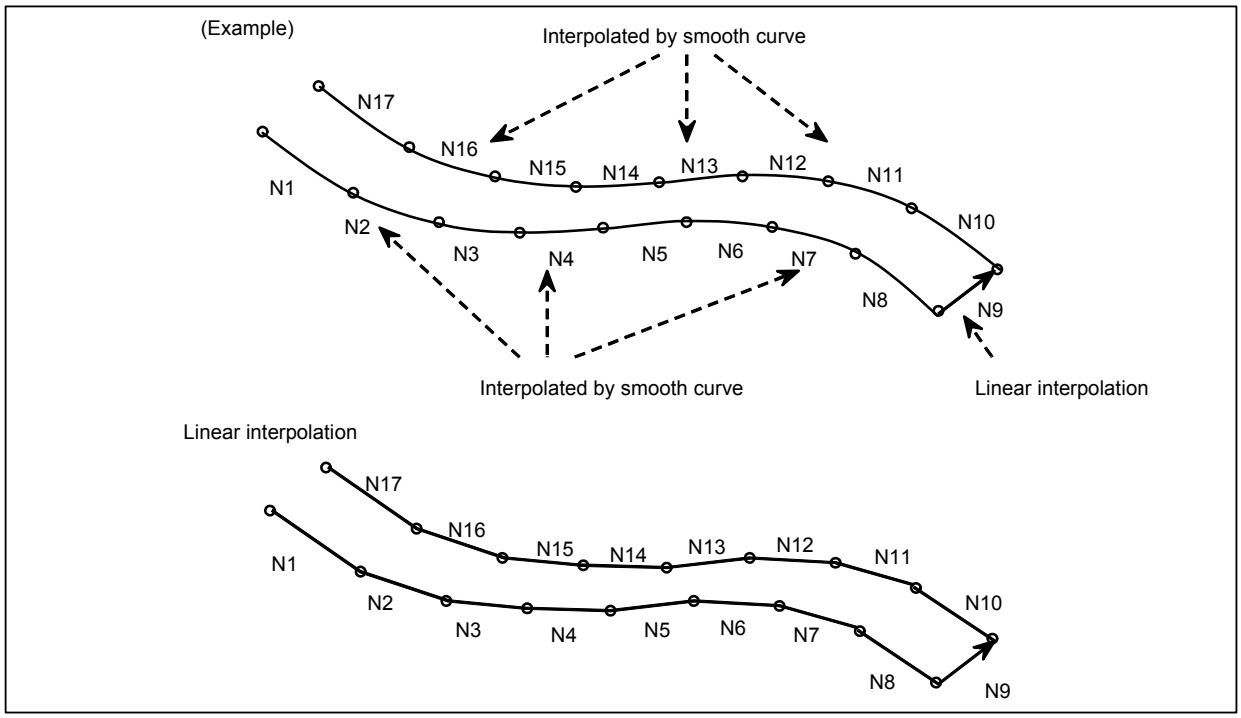
Profile	Portions having mainly a small radius of curvature	Portions having mainly a large radius of curvature

Example of machined parts	Automobile parts	Decorative parts, such as body side moldings
Length of line segment	Short	Long
Resulting surfaces produced using high-precision contour control	Smooth surface even when machining is performed exactly as specified by a program	Uneven surfaces may result when machining is performed exactly as specified by a program



Example of uneven surfaces (polygon) resulting from machining that precisely follows the line segments.

In smooth interpolation mode, the CNC automatically determines, according to the program command, whether an accurate figure is required, such as at corners, or a smooth figure is required where the radius of curvature is large. If a block specifies a travel distance or direction which differs greatly from that in the preceding block, smooth interpolation is not performed for that block. Linear interpolation is performed exactly as specified by the program command. Programming is thus very simple.



- Automatically turning on and off AI contour control with smooth interpolation

Specifying G5.1 Q2 also enables smooth interpolation and AI contour control to be turned on at the same time. The automatic velocity control by AI contour control reduces impacts on the mechanical system. Specifying G5.1 Q0 cancels the smooth interpolation mode and the AI contour control mode at the same time.

- Conditions for enabling smooth interpolation

Smooth interpolation is performed when all the following conditions are satisfied. If any of the following conditions is not satisfied for a block, that block is executed without smooth interpolation then the conditions are checked for the next block.

(1) The machining length specified in the block is shorter than the length specified with parameter No. 8486.
(2) The machining length specified in the block is longer than the length specified with parameter No. 8490.
(3) The difference in angle between the specified blocks is smaller than the value specified with parameter No. 8487.
(4) The machining length is other than 0.
(5) The modes are:
  G01 : Linear interpolation
  G13.1 : Polar coordinate interpolation cancel
  G15 : Polar coordinate command cancel
  G40 : Tool radius/tool nose radius compensation cancel (except for 3-dimensional cutter compensation)
  G64 : Cutting mode
  G80 : Canned cycle cancel
  G94 : Feed per minute
(6) Machining is specified only along the axes specified with G05.1Q2.
(7) The block is judged to be suitable for smooth interpolation, as performed with the internal algorithm of the CNC.

- Commands which cause alarm in smooth interpolation mode

When the following commands are commanded in smooth interpolation mode, alarm PS5110 (IMPROPER G-CODE(AICC MODE)) is issued.

(1) G27 : Reference position return check
(2) G28 : Automatic return to reference position
(3) G29 : Movement from reference position
(4) G30 : 2nd, 3rd, and 4th reference position return
(5) G30.1 : Floating reference position return

When the following commands are commanded in smooth interpolation mode, alarm PS5421 (ILLEGAL COMMAND IN G43.4/G43.5) is issued.

(1) G43.4, G43.5 : Tool center point control
(2) G43.4, G43.5 : Tool posture control
(3) G43.8, G43.9 : Cutting point commands

- Checking the smooth interpolation mode

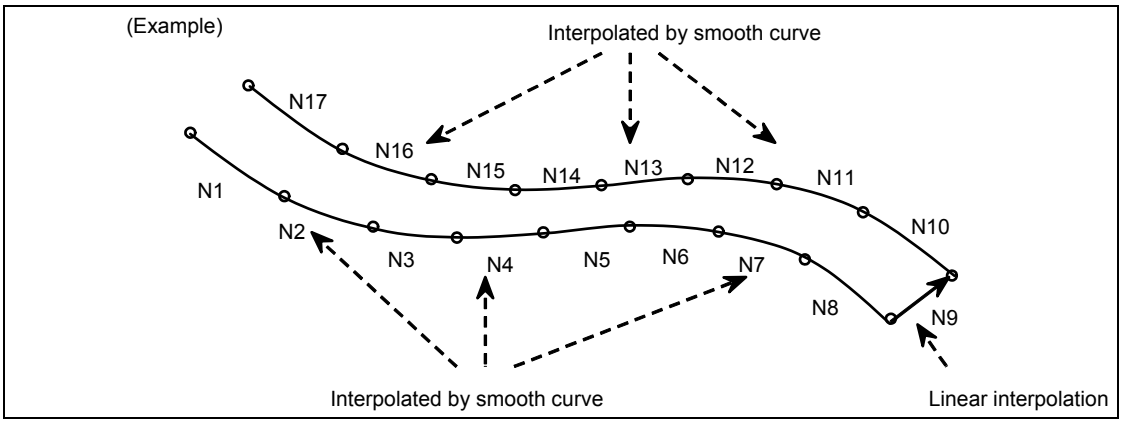
Diagnostic data No. 5000#0 indicates whether the smooth interpolation mode is enabled in the current block.
If the smooth interpolation mode is enabled, "smooth interpolation on" bit is set to 1.

Example

<Example program for smooth interpolation>

```
    .
    .
    .
G91   ;
G05.1  Q2   X0   Y0   Z0   ;
N01   G01   X1000   Z-300   ;
N02   X1000   Z-200   ;
N03   X1000   Z-50   ;
N10   X-1000   Z350   ;
N11   X-1000   Z175   ;
N12   X-1000   Z25   ;
N13   X-1000   Z-50   ;
N14   X-1000   Z-50   ;
N15   X-1000   Z50   ;
N16   X-1000   Z200   ;
N17   X-1000   Z300   ;
```

```
N04  X1000  Z50   ;                G05. 1  Q0  ;
N05  X1000  Z50   ;                   .
N06  X1000  Z-25  ;                   .
N07  X1000  Z-175 ;
N08  X1000  Z-350 ;
N09  Y1000  ;
```



Limitation

Basically, limitations on AI contour control apply.
In addition, the limitations below also apply.

- Controlled axes

Smooth interpolation can be specified only for the X-, Y-, and Z-axes and any axes parallel to these axes (up to three axes at one time).

- Modal G codes usable when smooth interpolation is specified

In a modal G code state listed below, smooth interpolation can be specified.
Do not specify smooth interpolation in modal states other than these.

G13.1 : Polar coordinate interpolation mode cancel
G15 : Polar coordinate command cancel
G40 : Tool radius compensation cancel
G40.1 : Normal direction control cancel
G49(G49.1),G43,G44 : Tool length compensation cancel or tool length compensation
G50 : Scaling cancel
G50.1 : Programmable mirror image cancel
G50.2 : Polygon turning cancel
G54.2P0 : Rotary table dynamic fixture offset cancel
G54.4P0 : Workpiece setting error compensation cancel
G64 : Cutting mode cancel
G67 : Macro modal call cancel
G69 : Coordinate system rotation/3-dimensional coordinate system conversion cancel
G80 : Canned cycle cancel
G94 : Feed per minute
G97 : Constant surface speed control cancel

- Unavailable operations

In smooth interpolation mode, do not perform the operations below.

- Manual intervention
- Manual handle retrace
- Program restart (excluding a program restart using a block number)

- Tool retract and recover
- Retrace
- Active block cancel
- Interruption type custom macro

- Functions that cannot be used together with smooth interpolation

Smooth interpolation cannot be used together with the functions below.

- Parallel axis control
- Twin table control

- Tool length compensation

Specify tool length compensation before specifying smooth interpolation. Do not change the amount of compensation in smooth interpolation mode.

4.13 NANO SMOOTHING

Overview

When a desired sculptured surface is approximated by minute segments, the nano smoothing function generates a smooth curve inferred from the programmed segments and performs necessary interpolation.
The nano smoothing function infers a curve from a programmed figure approximated with segments within tolerance. If the spacing between adjacent inflection points or programmed points is not constant, this function can generate a smoother curve than the conventional smooth interpolation function.
The interpolation of the curve reduces the segment approximation error, and the nano-interpolation makes the cutting surface smoother.
The interpolation of the curve reduces the segment approximation error, and the nano interpolation makes the cutting surface smoother.
Nano smoothing 2 allows the basic three axes (X, Y, and Z) or their parallel axes as well as two rotation axes to be specified. This enables five-axis machining (including rotation axes) to produce very smooth finished surfaces and so finds applications such as machining of smooth workpiece side faces.
In addition, this function can be used with tool center point control (type 1 (G43.4)). This eliminates the necessity of modifying a program when the tool length is changed.
For this function, the AI contour control option is required.
For nano smoothing 2, the nano smoothing option is required.

Format

G5.1 Q3 Xp0 Yp0 Zp0 [α0] [β0] ; : Nano smoothing mode on
G5.1 Q0 ; : Nano smoothing mode off

Xp : X-axis or an axis parallel to the X-axis
Yp : Y-axis or an axis parallel to the Y-axis
Zp : Z-axis or an axis parallel to the Z-axis
α, β : Rotary axis

NOTE
1 Specify G5.1 alone in a block.
(Avoid specifying any other G code in the same block.)
2 Specify position 0 for the axis programmed in the nano smoothing mode on block. The specified axis is subjected to nano smoothing, but no movement is made even in the absolute programming mode.
(Axis moving is not performed in the G05.1Q3 block.)
3 Nano smoothing mode is also turned off at a reset.

In the G5.1 Q3 block, specify the axis subject to nano smoothing. Note that up to three axes can be subject to the nano smoothing command at a time and that only the following axes can be specified.

- Basic three axes (X,Y,Z)
- Axes parallel to the basic three axes

For nano smoothing 2, up to two rotation axes can be specified in addition to the axes above.
If specifying the machining condition selecting function, specify G5.1 Q1 Rx first and then nano smoothing.

Example

```
O0010                          If the following functions are required
…                              before nano smoothing, specify G5.1.
(G5.1 Q1 R1;) ———————————— - AI contour control
G5.1 Q3 X0 Y0 Z0;              - Machining condition selecting function
…
G5.1 Q0; ———————————————— Nano smoothing mode off
                               AI contour control mode off
…
M30;
```

- Using nano smoothing 2 together with tool center point control

If using nano smoothing 2 together with tool center point control (type 1 (G43.4)), enable tool center point control (G43.44) first, and then enable nano smoothing 2 (G5.1 Q3).
To cancel them, cancel nano smoothing 2 (G5.1 Q0) first, and then cancel tool center point control (G49).

Example

```
O0010
…
G43.4 H1 P0;
G5.1 Q3 X0 Y0 Z0 B0 C0;
…
G5.1 Q0;
G49;
…
M30;
```

Tool center point control

Nano smoothing 2

- Switching between nano smoothing and nano smoothing 2

During nano smoothing, nano smoothing 2 cannot be specified.
(Similarly, during nano smoothing 2, nano smoothing cannot be specified.)
Switching between nano smoothing and nano smoothing 2 requires that G5.1 Q0 be specified.

Example: Switching from nano smoothing to nano smoothing 2

```
O0010
…
G5.1 Q3 X0 Y0 Z0;
                        } Nano smoothing
G5.1 Q0;                  (for 3-axis machining)
G5.1 Q3 X0 Y0 Z0 A0 B0;
…                       } Nano smoothing 2
G5.1 Q0;                  (for 5-axis machining)
…
M30;
```

Explanation

Generally, a program approximates a sculptured surface with minute segments with a tolerance of about 10 μm.

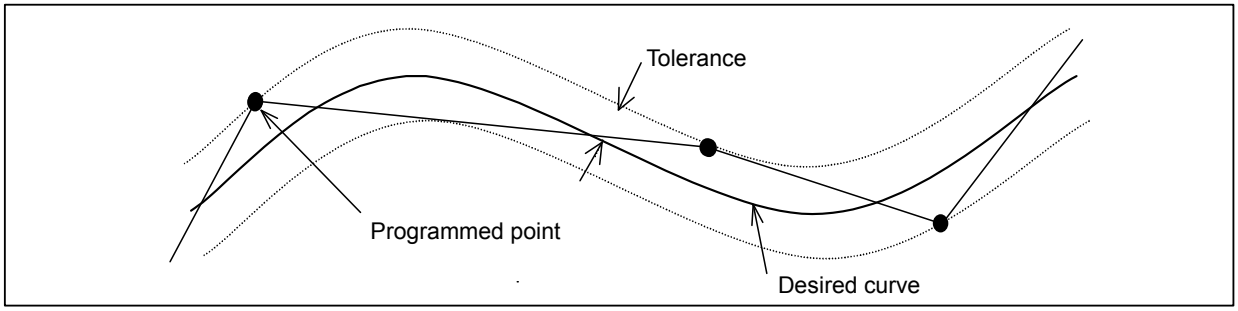


Fig. 4.13 (a)

Many programmed points are placed on the boundary of tolerance. The programmed points also have a rounding error owing to the least input increment of the CNC. The nano smoothing function creates multiple insertion points between adjacent programmed points so that a smooth curve can be created from the approximation segments. The desired curve is inferred from the insertion points of multiple blocks including buffered blocks.

Many insertion points are closer to the desired curve than the programmed points. A stable curve can be inferred with the insertion points created from multiple blocks including buffered blocks. Because the position of each insertion point is corrected in a unit smaller than the least input increment of the CNC within tolerance, the impact of rounding error is reduced.
Nano-interpolation is performed for the curve inferred from the corrected insertion points, so the resultant cutting surface becomes smooth.

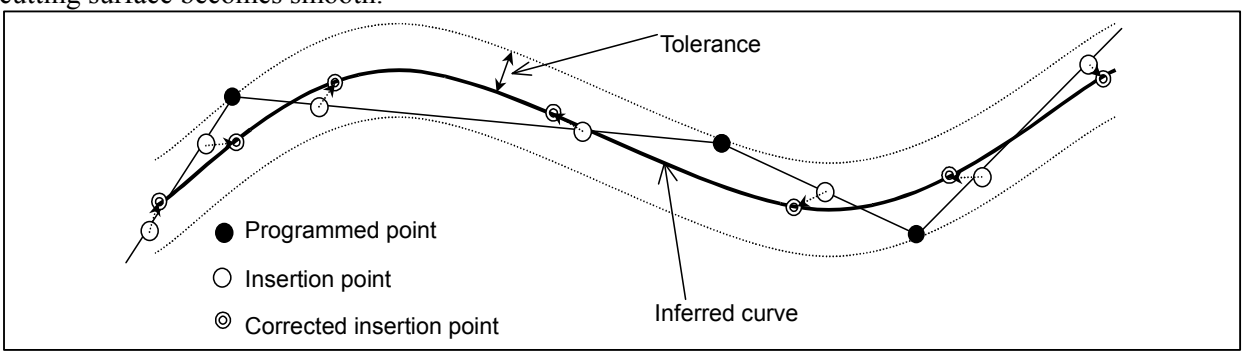


Fig. 4.13 (b)

- Specifying the tolerance

The tolerance of the program of nano smoothing is specified in parameter No. 19581.
The insertion points are corrected within tolerance, and a curve is inferred accordingly.

If 0 is specified in parameter No. 19581, the minimum travel distance in the increment system is considered to be the tolerance.

- Nano smoothing 2

Nano smoothing 2 performs smooth interpolation for the basic three axes (or their parallel axes) and two rotation axes independently.

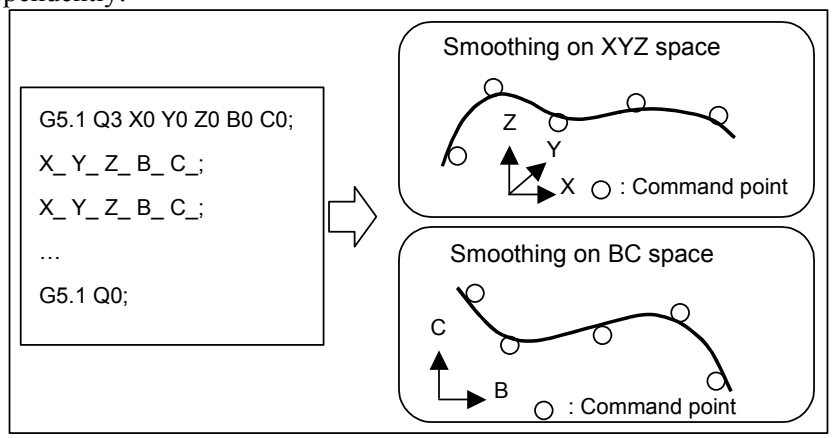


Fig. 4.13 (c)

Insertion points for the rotation axes are corrected so that each axis element of the distance between an insertion point and the corrected insertion point for both axes does not exceed the tolerance set for each axis in parameter No. 19587.

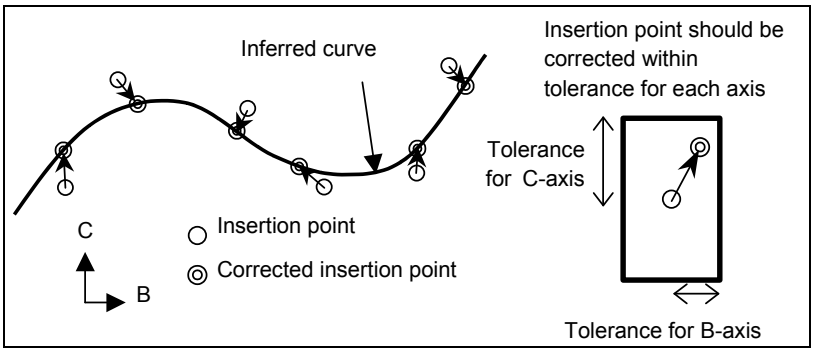


Fig. 4.13 (d)

- Making a decision on the basis of the spacing between adjacent programmed points

If the spacing between adjacent programmed points (block length) exceeds the value specified in parameter No. 8486 or falls below the value specified in parameter No. 8490 in the nano smoothing mode, the nano smoothing mode is cancelled at the start point of the block. Linear interpolation can be performed in the block.

When a decision is made on the basis of the spacing between adjacent programmed points, only the basic three axes (or their parallel axes) are considered, and the rotation axes are excluded. When the nano smoothing mode is canceled in a block, nano smoothing for the rotation axes is not performed, either.

If the values specified in the parameters are 0, no decision is made on the basis of the spacing between adjacent programmed points.

- Making a decision at a corner

If the difference in angle (see the Fig. 4.13 (e)) between adjacent programmed blocks exceeds the value specified in parameter No. 8487 in the nano smoothing mode, the nano smoothing mode is cancelled at the corner.

The decision at the corner is made by considering the basic three axes (or their parallel axes) only; the rotation axes are not considered. When the nano smoothing mode is canceled in a block, nano smoothing for the rotation axes is not performed, either.

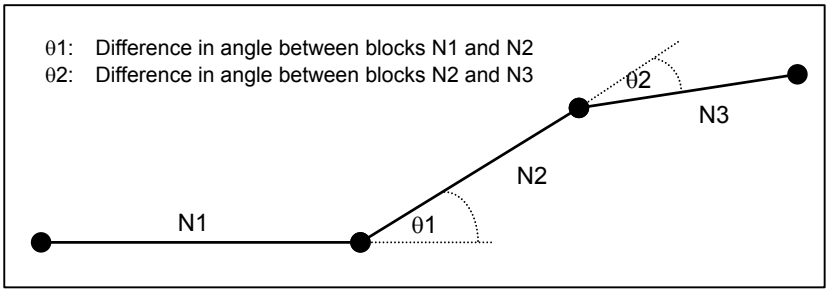


Fig. 4.13 (e)

If the value specified in the parameter is 0, no decision is made at the corner on the basis of the difference in angle.
Very minute blocks created for some reasons such as a calculation error of CAM can be ignored, and a smooth connection can be made at a corner. To do this, specify parameter No. 19582 to the minimum travel distance with which a decision is made on the basis of difference in angle. Then, the decision at a corner is disabled for a block of which distance is less than the specified minimum travel distance.

However, a decision based on the spacing between adjacent programmed points specified in parameter No. 8490 has higher priority than the decision at a corner. Therefore, the value specified in parameter No. 19582 must be greater than the value specified in parameter No. 8490.

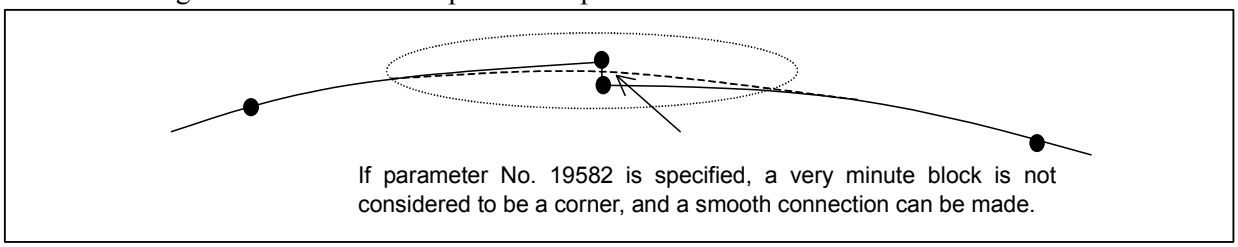


Fig. 4.13 (f)

- Automatically turning on and off AI contour control with nano smoothing

Specifying G5.1 Q3 also enables nano smoothing and AI contour control to be turned on at the same time. The automatic velocity control by AI contour control reduces impacts on the mechanical system. Specifying G5.1 Q0 cancels the nano smoothing and the AI contour control mode at the same time.

- Conditions for enabling nano smoothing

Nano smoothing is enabled if the conditions below are satisfied.
In a block that does not satisfy the conditions for enabling it, nano smoothing is canceled, and it is judged in the next block whether to perform nano smoothing anew.
In the following description, "block length" and "angle difference between blocks" apply to the basic three axes (or axes parallel to them) only, not rotation axes. Note, however, that in a block in which nano smoothing mode is canceled due to any of these conditions, nano smoothing on rotation axes will not be performed, either.

(1) The specified block length is less than parameter No. 8486.
(2) The specified block length is greater than parameter No. 8490.
(3) The angle difference between the specified blocks is less than parameter No. 8487.
(4) The mode is one of the following:
- Linear interpolation
- Feed per minute or inverse time feed
- Tool radius compensation cancel
- Canned cycle cancel
- Scaling cancel
- Macro modal call cancel
- Constant surface speed control cancel

- Cutting mode
- Coordinate system rotation/3-dimensional coordinate system conversion cancel
- Polar coordinate command cancel
- Normal direction control cancel
- Polar coordinate interpolation cancel
- Programmable mirror image cancel
- Polygon turning cancel

(5) The block does not contain a one shot G code command.
(6) The block does not suppress look ahead (buffering).
(7) The block contains a move command for only an axis subject to nano smoothing.

- Checking the nano smoothing

Diagnostic data (No. 5000) indicates whether the nano smoothing mode is enabled in the current block.
If the nano smoothing mode is enabled, "smoothing on" bit is set to 1.

Limitation

- Modal G codes usable when nano smoothing is specified

In a modal G code state listed below, nano smoothing can be specified.
Do not specify smooth interpolation in modal states other than these.

G13.1 : Polar coordinate interpolation mode cancel
G15 : Polar coordinate command cancel
G40 : Tool radius compensation cancel
G40.1 : Normal direction control cancel
G49(G49.1),G43,G44 : Tool length compensation cancel or tool length compensation
G50 : Scaling cancel
G50.1 : Programmable mirror image cancel
G50.2 : Polygon turning cancel
G54.2P0 : Rotary table dynamic fixture offset cancel
G54.4P0 : Workpiece setting error compensation cancel
G64 : Cutting mode cancel
G67 : Macro modal call cancel
G69 : Coordinate system rotation/3-dimensional coordinate system conversion cancel
G80 : Canned cycle cancel
G94,G93 : Feed per minute or inverse time feed
G97 : Constant surface speed control cancel

- Single-block operation

When single-block operation is carried out in the nano smoothing mode, the operation stops at a corrected insertion point not at a programmed point.
Even in the nano smoothing mode, normal single-block operation is carried out for a block that does not satisfy the conditions of nano smoothing mode.

- Tool length compensation

To carry out tool length compensation, specify the command before specifying nano smoothing. Avoid changing the amount of compensation in the nano smoothing mode.
If G43, G44, or G49 is specified in a block between the block in which the command of nano smoothing mode on (G5.1 Q3) is specified and the block in which the command of nano smoothing mode off (G5.1 Q0) is specified, an alarm PS0343 will be issued.

- Tool radius/tool nose radius compensation

If tool radius/tool nose radius compensation is specified in the nano smoothing mode, the nano smoothing mode is cancelled. Then, when the command of tool radius/tool nose radius compensation cancel (G40) is specified, a decision is made whether to start nano smoothing from the next block. The startup and cancel operations of type C are always carried out for the tool radius/tool nose radius compensation specified in the nano smoothing mode, irrespective of the parameter setting.
A command related to tool radius/tool nose radius compensation should not be specified in the nano smoothing mode unless it is absolutely necessary.

- Interruption type custom macro

No interruption type custom macro can be used in the nano smoothing mode.
If the nano smoothing mode is specified while an interruption type custom macro is enabled or if an interruption type custom macro is enabled in the nano smoothing mode, an alarm PS0342 will be issued.

- Manual intervention

Manual intervention by specifying the manual absolute on command cannot be performed in the nano smoothing mode. If this is attempted, an alarm PS0340 will be issued at the cycle start after manual intervention.

- Rotary table dynamic fixture offset

The command of rotary table dynamic fixture offset (G54.2) must be cancelled before specifying the nano smoothing mode. These commands cannot be used in the nano smoothing mode. If an attempt is made to use one of these commands, an alarm PS0343 will be issued.

- Number of blocks that can be specified successively

Up to about 300,000,000 blocks can be specified successively in the nano smoothing mode. If more blocks are specified, an alarm PS0341 will be issued.
However, when a block which does not satisfy the conditions of the nano smoothing mode is encountered, the mode is canceled and the counted number of successive blocks is reset to 0.

- Continuity of a program

Curve interpolation is carried out for multiple programmed blocks including buffered blocks in the nano smoothing mode.
Therefore, the programmed commands must be executed continuously in the nano smoothing mode.
The continuity of a program may be lost, and continuous execution may not be performed, in some cases such as the following: A single-block stop is made in the nano smoothing mode; and another program is executed in the MDI mode. If this occurs, an alarm PS0344 will be issued.

- Restrictions on resumption of automatic operation

(1) Resuming a program
Curve interpolation is performed for corrected insertion points not for programmed points in the nano smoothing mode. Accordingly, when a sequence number is specified to resume the program, the operation cannot be resumed from a programmed point in a block.
To resume a program, specify a block number, using the block counter displayed in the program screen.
(2) Retracting and returning the tool
The tool cannot be retracted or returned in the nano smoothing mode.
(3) Retracing
Retracing cannot be performed in the nano smoothing mode.
(4) Manual handle retrace
In nano smoothing mode, manual handle retrace cannot be performed.
(5) Active block cancel
The active block cancel function is temporarily disabled in the nano smoothing mode.

- Functions that cannot be used simultaneously

The nano smoothing function cannot be used simultaneously with the following functions.

- Parallel axis control
- Twin table control

- Background graphic display

The background graphic display function draws the path in the nano smoothing mode by linear interpolation.

- Tool center point control

Nano smoothing 2 can be used with tool center point control (type 1 (G43.4)). However, when it is used with tool center point control (type 2 (G43.5)) or tool posture control, alarm PS5421, “ILLEGAL COMMAND IN G43.4/G43.5” is issued.

When specifying nano smoothing 2 in the tool center point control (type 1 (G43.4)) mode, be sure to specify rotation axes as axes for nano smoothing. If only linear axes are specified, alarm PS5421, “ILLEGAL COMMAND IN G43.4/G43.5” is issued.

4.14 NURBS INTERPOLATION (G06.2)

Many computer-aided design (CAD) systems used to design metal dies for automobiles and airplanes utilize non-uniform rational B-spline (NURBS) to express a sculptured surface or curve for the metal dies.

This function enables NURBS curve expression to be directly specified to the CNC. This eliminates the need for approximating the NURBS curve with minute line segments. This offers the following advantages:

1. No error due to approximation of a NURBS curve by small line segments
2. Short part program
3. No break between blocks when small blocks are executed at high speed
4. No need for high-speed transfer from the host computer to the CNC

When this function is used, a computer-aided machining (CAM) system creates a NURBS curve according to the NURBS expression output from the CAD system, after compensating for the length of the tool holder, tool diameter, and other tool elements. The NURBS curve is programmed in the NC format by using these three defining parameters: control point, weight, and knot.

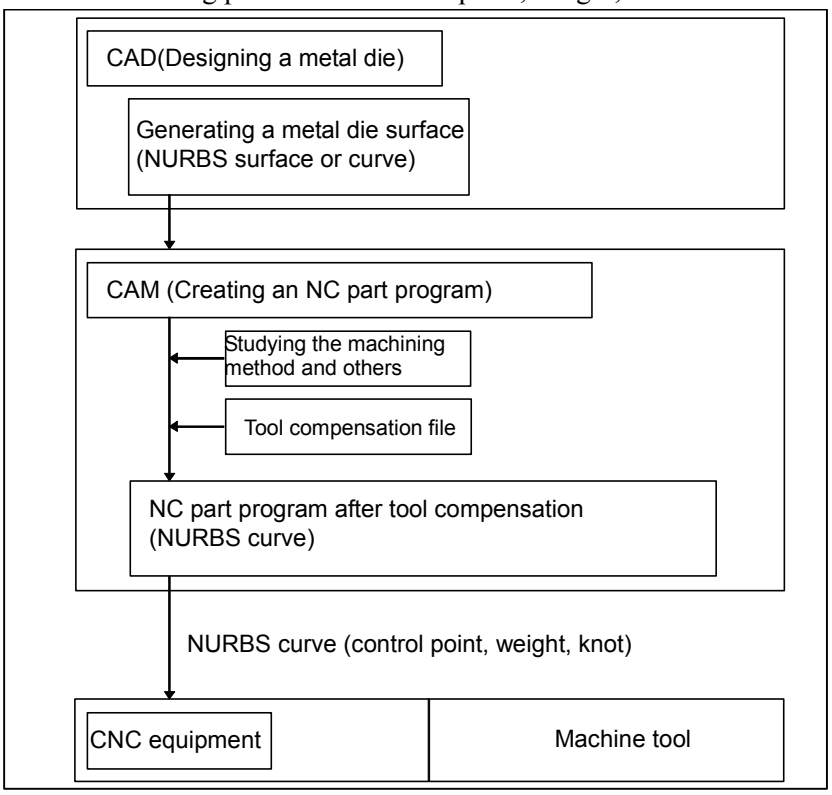


Fig. 4.14 (a) NC part program for machining a metal die according to a NURBS curve

NURBS interpolation can be performed for up to five axes (including two rotation axes). Therefore, NURBS interpolation can be performed for the basic three axes (X, Y, and Z) and two rotation axes at the same time. This enables five-axis machining to produce very smooth finished surfaces and so finds applications such as machining of smooth workpiece side faces.

In addition, NURBS interpolation can be used with tool center point control (type 1 (G43.4)). This eliminates the necessity of modifying a program when the tool length is changed.

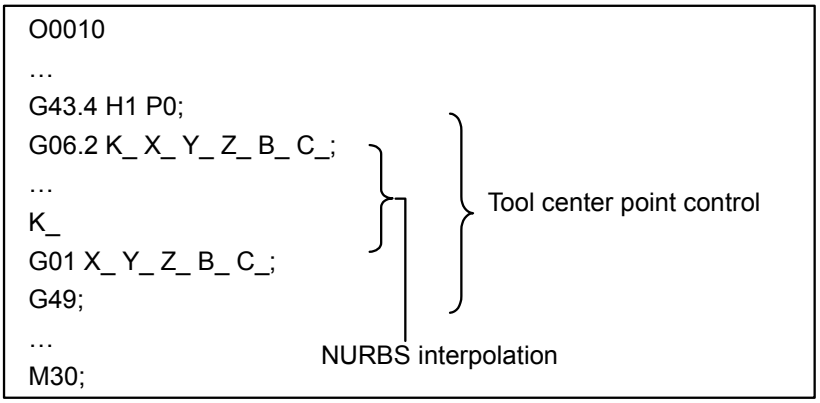
Format

```
G06.2[P_] K_X_Y_Z_[α_] [β_][R_] [F_];
          K_X_Y_Z_[α_] [β_][R_];
          K_X_Y_Z_[α_] [β_][R_];
          K_X_Y_Z_[α_] [β_][R_];
           :
          K_X_Y_Z_[α_] [β_][R_];
          K_;
          :
          K_;
G01 . . .
```

G06.2 : Start NURBS interpolation mode
P_ : Rank of NURBS curve
X_Y_Z_ : Control point
α_β_ : Control point (rotary axes)
R_ : Weight
K_ : Knot
F_ : Feedrate

If executing NURBS interpolation simultaneously with tool center point control (type 1 (G43.4)), enable tool center point control (G43.4) first and then enable NURBS interpolation (G06.2). To cancel them, end NURBS interpolation (G code command in group 01 other than G06.2) first and then cancel tool center point control (G49).

Example :



Explanation

- NURBS interpolation mode

NURBS interpolation mode is selected when G06.2 is programmed. G06.2 is a modal G code of group 01. NURBS interpolation mode ends when a G code of group 01 other than G06.2 (G00, G01, G02, G03, etc.) is specified.

- Rank of NURBS

A rank of NURBS can be specified with address P. The rank setting, if any, must be specified in the first block. If the rank setting is omitted, a rank of four (degree of three) is assumed for NURBS. The valid data range for P is 2 to 4. The P values have the following meanings:
P2: NURBS having a rank of two (degree of one)
P3: NURBS having a rank of three (degree of two)
P4: NURBS having a rank of four (degree of three) (default)

This rank is represented by k in the defining expression indicated in the description of NURBS curve below. For example, a NURBS curve having a rank of four has a degree of three. The NURBS curve can be expressed by the constants t^3, t^2, and t^1.

- Weight

The weight of a control point programmed in a single block can be defined. When the weight setting is omitted, a weight of 1.0 is assumed.

- Knot

The number of specified knots must equal the number of control points plus the rank value. In the blocks specifying the first to last control points, each control point and a knot are specified in an identical block. After these blocks, as many blocks (including only a knot) as the rank value are specified. The NURBS curve programmed for NURBS interpolation must start from the first control point and end at the last control point. The first k knots (where k is the rank) must have the same values as the last k knots (multiple knots). If the absolute coordinates of the start point of NURBS interpolation do not match the position of the first control point, alarm PS5117 is issued. (To specify incremental values, G06.2 X0 Y0 Z0 K_ must be programmed.)

- NURBS curve

Using these variables:

k : Rank

Pi : Control point

Wi : Weight

Xi : Knot ($X_i < X_{i+1}$)

Knot vector [$X_0, X_1, \ldots, X_m$] ($m = n + k$)

t : Spline parameter,

the spline basis function N can be expressed with the de Boor-Cox recursive formula, as indicated below:

$$N_{i,1}(t) = \begin{cases} 1 (x_i < t < 1 x_{i+1}) \\ 0 (t < x_i, x_{i+1} < t) \end{cases}$$

$$N_{i,k}(t) = \frac{(t - x_i) N_{i,k-1}(t)}{x_{i+k-1} - x_i} + \frac{(x_{i+k} - t) N_{i+1,k-1}(t)}{x_{i+k} - x_{i+1}}$$

The NURBS curve P(t) of interpolation can be expressed as follows:

$$P(t) = \frac{\sum_{i=0}^{n} N_{i,k}(t) w_i p_i}{\sum_{i=0}^{n} N_{i,k}(t) w_1}$$

$(X0 \leq t \leq Xm)$

- Reset

A reset during NURBS interpolation results in the clear state. The modal code of group 1 enters the state specified in bit 0 (G01) of parameter No. 3402.

Limitation

- Controlled axes

NURBS interpolation can be performed on up to five axes (including two rotary axes). The axes of NURBS interpolation must be specified in the first block. A new axis cannot be specified before the beginning of the next NURBS curve or before NURBS interpolation mode ends.

- Command in NURBS interpolation mode

In NURBS interpolation mode, any command other than the NURBS interpolation command (miscellaneous function and others) cannot be specified.

- Manual intervention

If manual intervention is attempted while manual absolute mode is set, alarm PS5118 is issued.

- Tool radius compensation

Tool radius compensation cannot be simultaneously executed. NURBS interpolation can only be specified after tool radius compensation has been canceled.

- 3-dimensional cutter compensation

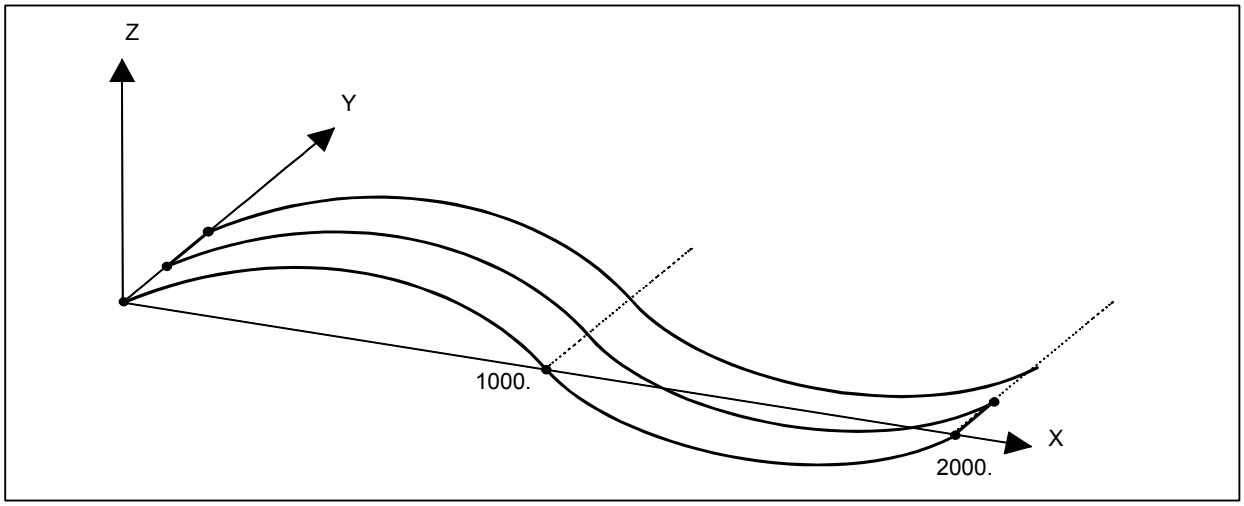
3-dimensional cutter compensation cannot be simultaneously executed. NURBS interpolation can only be specified after 3-dimensional cutter compensation has been canceled.

- Tool center point control

Tool center point control (type 1 (G43.4)) can be simultaneously executed, but tool center point control (type 2 (G43.5)) and tool posture control cannot be simultaneously executed. If an attempt is made to execute tool center point control (type 2 (G43.5) or tool posture control simultaneously with NURBS interpolation, alarm PS5421 is issued.

Example

```
<Sample NURBS interpolation program>
G90;
...
G06.2 K0.    X0.    Z0.;
      K0.    X300.  Z100.;
      K0.    X700.  Z100.;
      K0.    X1300. Z-100.;
      K0.5   X1700. Z-100.;
      K0.5   X2000. Z0.;
      K1.0;
      K1.0;
      K1.0;
      K1.0;
G01   Y0.5;
G06.2 K0.    X2000. Z0.;
      K0.    X1700. Z-100.;
      K0.    X1300. Z-100.;
      K0.    X700.  Z100.;
      K0.5   X300.  Z100.;
      K0.5   X0.    Z0.;
      K1.0;
      K1.0;
      K1.0;
      K1.0;
G01   Y0.5;
G06.2 ...
...
G01 ...
```



4.14.1 NURBS Interpolation Additional Functions

In the FANUC Series 30*i*/31*i*, NURBS interpolation provides the following additional functions:

- Parametric feedrate control

The maximum feedrate of each segment is determined according to a specified feedrate and acceleration. For a series of segments, the feedrates at the start point and end point of the segments are determined as described below. During a movement from the start point to end point, the feedrate changes continuously. This function can be used only for NURBS interpolation when bit 5 (FDI) of parameter No. 8412 is set to 1.

1. Start point speed
   (1) At the start of NURBS interpolation
   If the amount of travel in the previous block is 0, the maximum speed in the first segment is regarded as the start point speed. If the amount of travel in the previous block is not 0, whichever is smaller of the specified speed in the previous block and the maximum speed in the first segment is regarded as the start point speed.
   Previous block: Block immediately preceding the block in which NURBS interpolation is specified.
   (2) Second and subsequent segments
   Whichever is smaller of the maximum speed in the previous segment and the maximum speed in this segment is regarded as the start point speed.

2. End point speed
   (1) In a segment other than the last
   Whichever is smaller of the maximum speed in this segment and the maximum speed in the next segment is regarded as the end point speed.
   (2) At the end of NURBS interpolation
   If the amount of travel in the next block is 0, the maximum speed in this segment is regarded as the end point speed. If the amount of travel in the next block is not 0, whichever is smaller of the maximum speed in this segment and the specified speed in the next block is regarded as the end point speed.

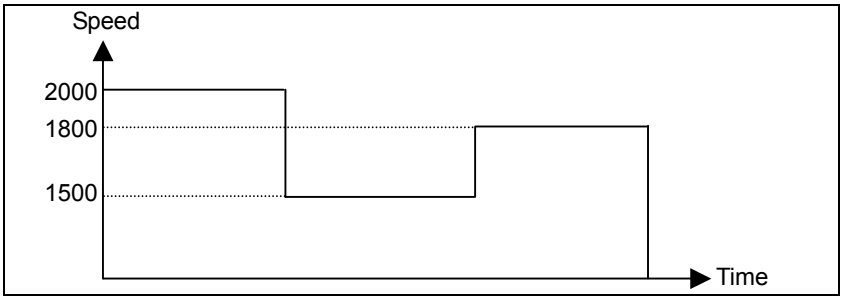
After the stop of a single block, the start point speed at a restart will be the maximum speed in this segment.

After the stop of a feed hold, the speed at a restart will be the same as the speed at the stop of the feed hold.

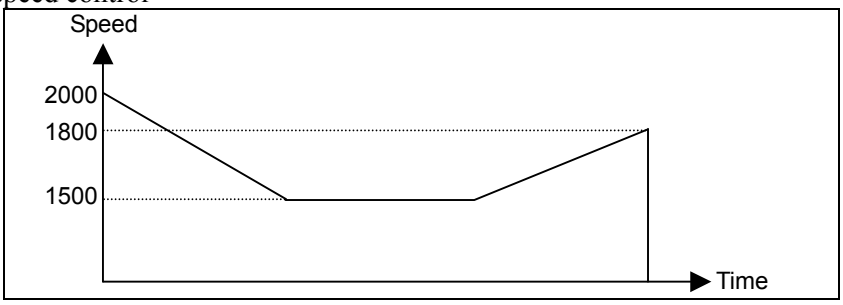
Example

1. Specified program
   G90 G06.2 X0. Y0. K0. F2000 ;
   X10. Y10. K0. F1500 ;
   X20. Y20. K0. F1800 ;
   X30. Y30. K0. ;
   X40. Y40. K1.
   X50. Y50. K2.
   K3.
   K3.
   K3.
   K3.

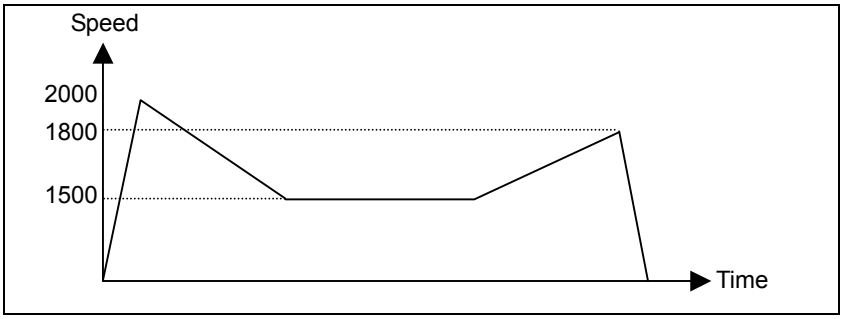
2. Specified speed



3. Parametric speed control



4. Because of the addition of acceleration/deceleration before interpolation, the actual cutting feedrate is as shown below.



NOTE
The actual acceleration/deceleration method is bell-shaped acceleration/deceleration.

- High-precision knot command

If bit 1 (HIK) of parameter No. 8412 is set to 1, knot commands with a whole number of up to 12 digits and a decimal fraction of up to 12 digits can be specified. This function can be used only for knot commands (address K) with a decimal point which are specified during NURBS interpolation.

If a high precision knot command is used, the knot command format is as shown below.

K (Number of A digits) . (Number of B digits)

where A + B must be 12 or less. The address K and the numbers including the decimal point must be 14 characters or less.

(Example)

Valid
K.999999999999
K1234.56789012
K999999999999.

Invalid (causing alarm PS0012 to be issued)
K0.999999999999
K1234.567891230
K1234.567891234
K999999999999.0

- Simple start command

If bit 0 (EST) of parameter No. 8412 is set to 1, a control command may be omitted at the first control point. Because the same value is set for the knot in the first block and the knot in the second block, the knot command may also be omitted in the first bock.

The format of a NURBS interpolation command using this function is as shown below.

```
G06.2 [P__ ] [K__ ] [IP__ ] [R__ ] [F__ ] ;
                K__ IP__ [R__ ] ;
                K__ IP__ [R__ ] ;
                K__ IP__ [R__ ] ;
                …
                K__ IP__ [R__ ] ;
                K__ ;
                …
                K__ ;
G01…
…
```

G06.2 : NURBS interpolation mode ON
P : Rank of the NURBS curve
IP : Control point
R : Weight
K : Knot
F : Speed

Words enclosed in [ ] are optional.

If a control point command is omitted in the first block, the current position as determined when NURBS interpolation is started is regarded as the first control point. Also, only the axis specified for the control point in the second block is handled as a NURBS interpolation axis and, therefore, in the second block, specify all axes subject to NURBS interpolation.

If a knot command is omitted in the first block, the knot is regarded as the same as the knot specified in the second block.

4.15 HYPOTHETICAL AXIS INTERPOLATION (G07)

In helical interpolation, when pulses are distributed with one of the circular interpolation axes set to a hypothetical axis, sine interpolation is enabled.
When one of the circular interpolation axes is set to a hypothetical axis, pulse distribution causes the speed of movement along the remaining axis to change sinusoidally. If the major axis for threading (the axis along which the machine travels the longest distance) is set to a hypothetical axis, threading with a fractional lead is enabled. The axis to be set as the hypothetical axis is specified with G07.

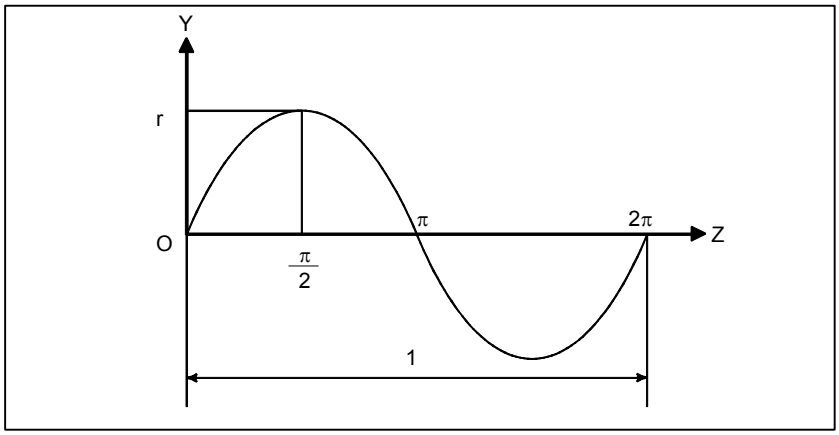
Format

G07 α0 ; Hypothetical axis setting
:
G07 α1 ; Hypothetical axis cancel
Where, α is any one of the addresses of the controlled axes.

Explanation

- Sine interpolation

The a axis is regarded as a hypothetical axis for the period of time from the G07 α0 command until the G07 α1 command appears.
Suppose sine interpolation is performed for one cycle in the YZ plane. The hypothetical axis is then the X axis.
$X^2+Y^2 = r^2$ (r is the radius of an arc.)
$Y = rSIN(\frac{2\pi}{1}Z)$ (1 is the distance traveled along the Z-axis in one cycle.)



- Interlock, stroke limit, and external deceleration

Interlock, stroke limit, and external deceleration can also apply to the hypothetical axis.

- Handle interruption

An interrupt caused by the handle also applies to the hypothetical axis. This means that movement for a handle interrupt is performed.

Limitation

- Manual operation

The hypothetical axis can be used only in automatic operation. In manual operation, it is not used, and movement takes place.

- Move command

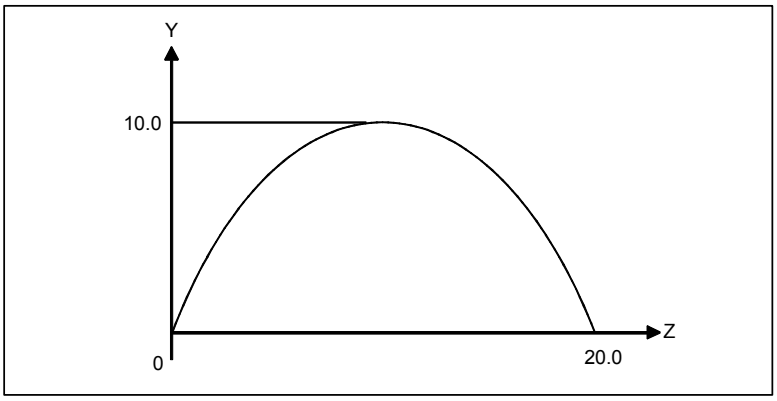
Specify hypothetical axis interpolation only in the incremental mode.

- Coordinate rotation

Hypothetical axis interpolation does not support coordinate rotation.

Example

- Sine interpolation



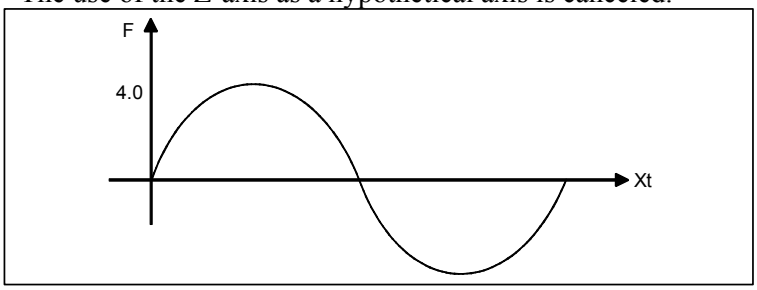

```
N001  G07  X0  ;
N002  G91  G17  G03  X-20.0  Y0.0  I-10.0  Z20.0  F100 ;
N003  G01  X10.0 ;
N004  G07  X1 ;
```

From the N002 to N003 blocks, the X-axis is set to a hypothetical axis. The N002 block specifies helical cutting in which the Z-axis is the linear axis. Since no movement takes place along the X axis, movement along the Y-axis is performed while performing sine interpolation along the Z-axis.
In the N003 block, there is no movement along the X-axis, and so the machine dwells until interpolation terminates.

- Changing the feedrate to form a sine curve

(Sample program)

```
G07Z0 ;              The Z-axis is set to a hypothetical axis.
G02X0Z0I10.0F4. ;    The feedrate on the X-axis changes sinusoidally.
G07Z1 ;              The use of the Z-axis as a hypothetical axis is canceled.
```



4.16 VARIABLE LEAD THREADING (G34)

Specifying an increment or a decrement value for a lead per screw revolution enables variable lead threading to be performed.

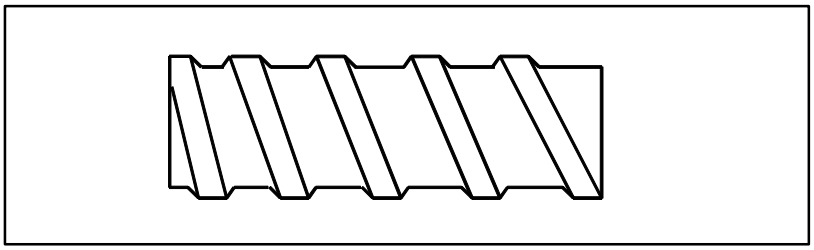


Fig. 4.16 (a) Variable lead screw

Format

G34 IP_ F_ K_ Q_ ;

IP_ : End point
F_ : Lead in longitudinal axis direction at the start point
K_ : Increment and decrement of lead per spindle revolution
Q_ : Shift amount of starting angle of thread cutting

Explanation

Address other than K are the same as in straight/taper thread cutting with G32.
The K value depends on the increment system of the reference axis, as indicated in Table 4.16 (a).
If the specified K value exceeds the range indicated in Table 4.16 (a), if the maximum lead is exceeded after a change due to the K value, or if the lead value is negative, an alarm PS0313 will be issued.

Table 4.16 (a) Range of valid K values

Increment system of reference axis	Metric input (mm/rev)			Inch input (inch/rev)		
IS-A	±0.001	to	±500.000	±0.00001	to	±50.00000
IS-B	±0.0001	to	±500.0000	±0.000001	to	±50.000000
IS-C	±0.00001	to	±50.00000	±0.0000001	to	±5.0000000
IS-D	±0.000001	to	±5.000000	±0.00000001	to	±0.50000000
IS-E	±0.0000001	to	±0.5000000	±0.000000001	to	±0.050000000

⚠ CAUTION
The "thread cutting cycle retract" is not effective for G34.

Example

Lead at the start point: 8.0 mm
Lead increment: 0.3 mm/rev

```
G34 Z-72.0 F8.0 K0.3 ;
```

4.17 CIRCULAR THREADING (G35, G36)

Using the G35 and G36 commands, a circular thread, having the specified lead in the direction of the major axis, can be machined.

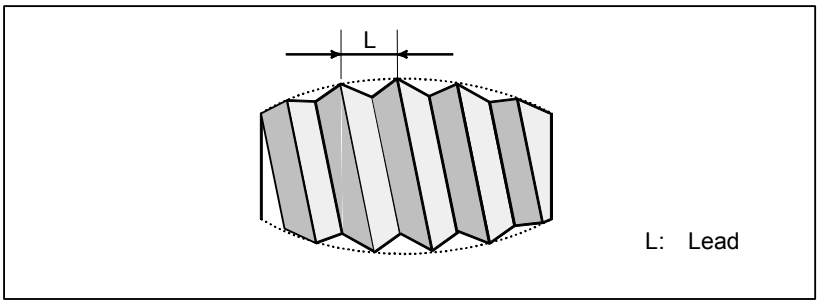


Fig. 4.17 (a) Circular threading

Format

A sample format for the G18 plane (Z-X plane) is indicated below. When using the format for the G17 plane (X-Y plane), change the addresses Z, X, K, and I to X, Y, I, and J respectively. When using the format for the G19 plane (Y-Z plane), change the addresses Z, X, K, and I to Y, Z, J, and K respectively.

M

$$\begin{Bmatrix} G35 \\ G36 \end{Bmatrix} X_\ Z_ \begin{Bmatrix} I_\ K_ \\ R_ \end{Bmatrix} F_\ Q_\ ;$$

G35 : Clockwise circular threading command
G36 : Counterclockwise circular threading command

X, Z : Specify the arc end point (in the same way as for G02, G03).
I, K : Specify the arc center relative to the start point, using relative coordinates (in the same way as for G02, G03).
R : Specify the arc radius.
F : Specify the lead in the direction of the major axis.
Q : Specify the shift of the threading start angle
(0° to 360°, with least input increment of 0.001)
(The value can be programmed with a decimal point.)

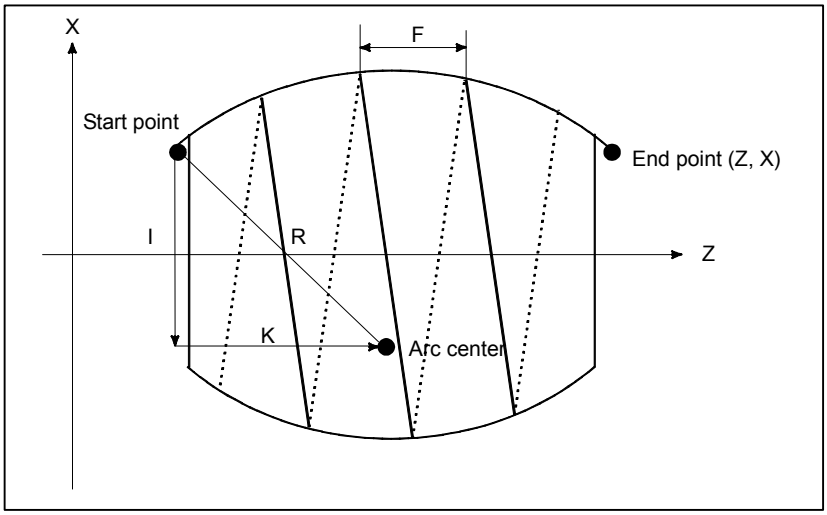
T

$$\begin{Bmatrix} G35 \\ G36 \end{Bmatrix} X(U)_\ Z(W)_ \begin{Bmatrix} I_\ K_ \\ R_ \end{Bmatrix} F_\ Q_\ ;$$

G35 : Clockwise circular threading command
G36 : Counterclockwise circular threading command

X(U), Z(W) : Specify the arc end point (in the same way as for G02, G03).

I, K : Specify the arc center relative to the start point, using relative coordinates (in the same way as for G02, G03).
R : Specify the arc radius.
F : Specify the lead in the direction of the major axis.
Q : Specify the shift of the threading start angle
(0° to 360°, with least input increment of 0.001)
(The value cannot be programmed with a decimal point.)



Explanation

- Specifying the arc radius

If R is specified with I and K, only R is effective.

- Shift angle

If an angle greater than 360° is programmed, it is set to 360°.

M

- Specifying shift angle Q

To specify the shift angle Q, set bit 0 (GQS) of parameter No. 3451 to 1.

T

- Automatic tool compensation

The G36 command is used to specify the following two functions: Automatic tool compensation X and counterclockwise circular threading. The function for which G36 is to be used depends on bit 3 (G36) of parameter No. 3405.

- When parameter G36 is set to 0, the G36 command is used for automatic tool compensation X.
- When parameter G36 is set to 1, the G36 command is used for counterclockwise circular threading.

G37.1 can be used to specify automatic tool compensation X and G37.2 can be used to specify automatic tool compensation Z.

(Specification method)
G37.1 X_
G37.2 Z_

- G code when bit 3 (G36) of parameter No. 3405 is set to 1

G code	G code group	Function
G35	01	Clockwise circular threading
G36		Counterclockwise circular threading
G37	00	Automatic tool compensation Z
G37.1		Automatic tool compensation X
G37.2		Automatic tool compensation Z

Limitation

- Range of specifiable arc

An arc must be specified such that it falls within a range in which the major axis of the arc is always the Z-axis or always the X-axis, as shown in Fig. 4.17 (b) and Fig. 4.17 (c). If the arc includes a point at which the major axis changes from the X-axis to Z-axis, or vice versa, as shown in Fig. 4.17 (d), an alarm PS5058 is issued.

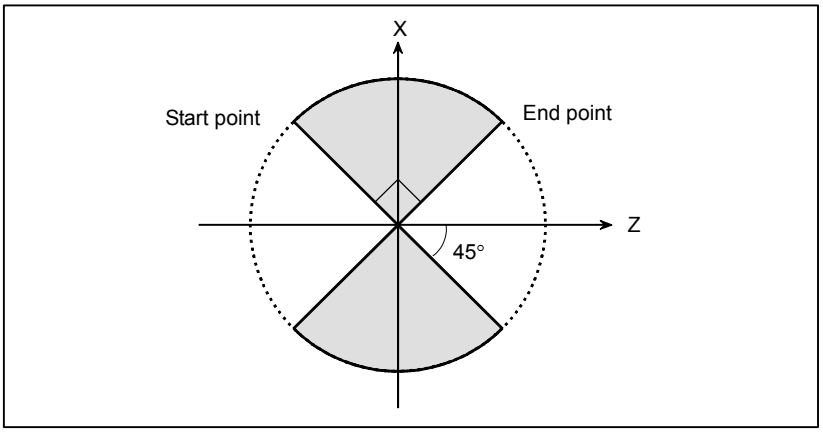


Fig. 4.17 (b) Range in which the Z-axis is the major axis

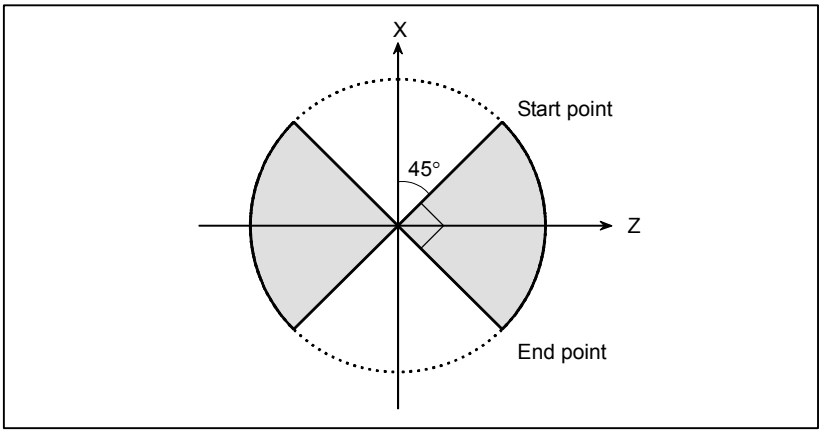


Fig. 4.17 (c) Range in which the X-axis is the major axis

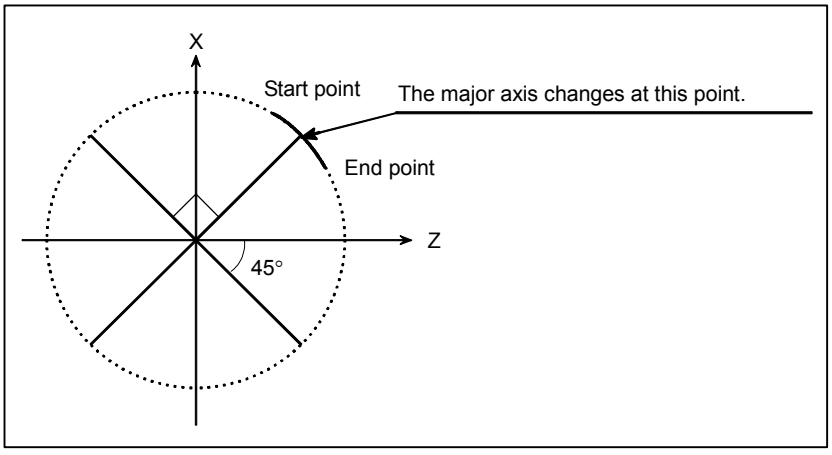


Fig. 4.17 (d) Example of arc specification which causes an alarm

- End point not on an arc

If the end point is not on an arc, a movement on an axis is made to a position of which coordinate matches the corresponding coordinate of the end point. Then, a movement is made on another axis to reach the end point.

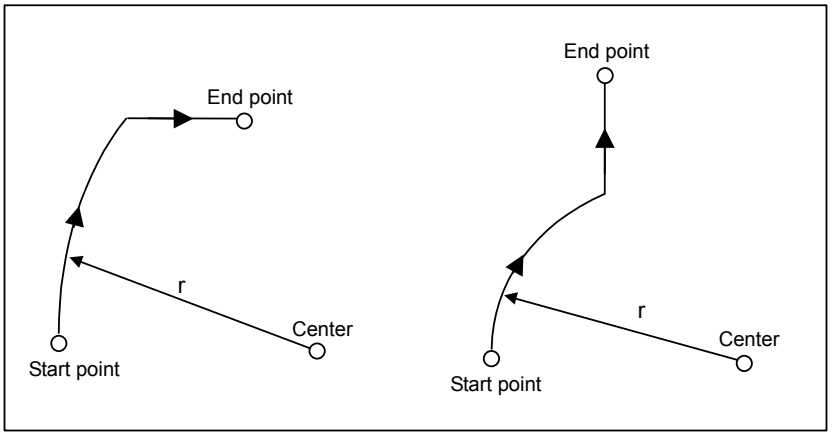


Fig. 4.17 (e) Movement when the end point is not on an arc

4.18 SKIP FUNCTION (G31)

Linear interpolation can be commanded by specifying axial move following the G31 command, like G01. If an external skip signal is input during the execution of this command, execution of the command is interrupted and the next block is executed.
The skip function is used when the end of machining is not programmed but specified with a signal from the machine, for example, in grinding. It is used also for measuring the dimensions of a workpiece.

Format

G31 IP ;

G31 : One-shot G code (If is effective only in the block in which it is specified)

Explanation

The coordinate values when the skip signal is turned on can be used in a custom macro because they are stored in the custom macro system variable #100151 to #100182, as follows. For systems with up to 20 axes, FS16-compatible system variables (#5061 to #5080) may also be used.

#100151 (#5061) : First axis coordinate value
#100152 (#5062) : Second axis coordinate value
:
#100170 (#5080) : 20th axis coordinate value

#100171 : 21st axis coordinate value
#100172 : 22nd axis coordinate value
:
#100182 : 32nd axis coordinate value

⚠ CAUTION

Disable feedrate override, dry run, and automatic acceleration/deceleration (however, these become available by setting bit 7 (SKF) of parameter No.6200 to 1.) when the feedrate per minute is specified, allowing for an error in the position of the tool when a skip signal is input. These functions are enabled when the feedrate per rotation is specified.

NOTE

If G31 command is issued while tool radius/tool nose radius compensation is applied, an alarm PS0035 is displayed. Cancel the tool radius compensation with the G40 command before the G31 command is specified.

Example

- **The next block to G31 is an incremental programming**

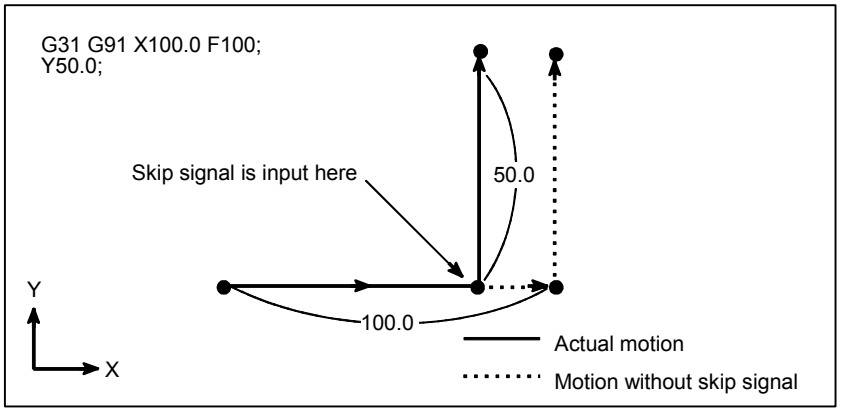


Fig. 4.18 (a) The next block is an incremental programming

- **The next block to G31 is an absolute programming for 1 axis**

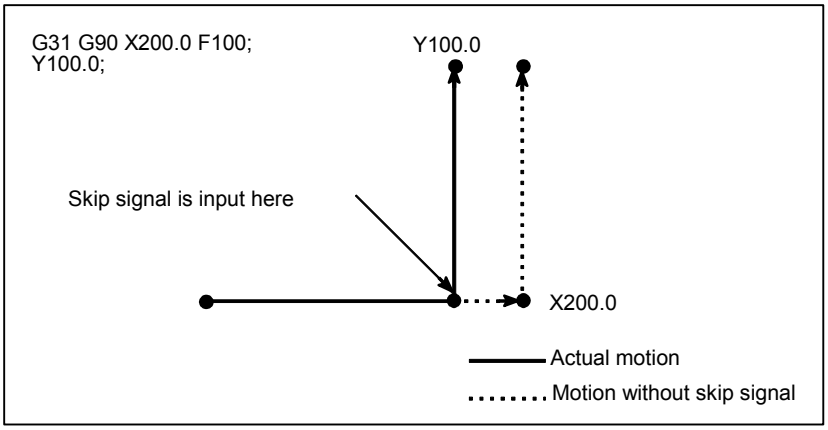


Fig. 4.18 (b) The next block is an absolute programming for 1 axis

- **The next block to G31 is an absolute programming for 2 axes**

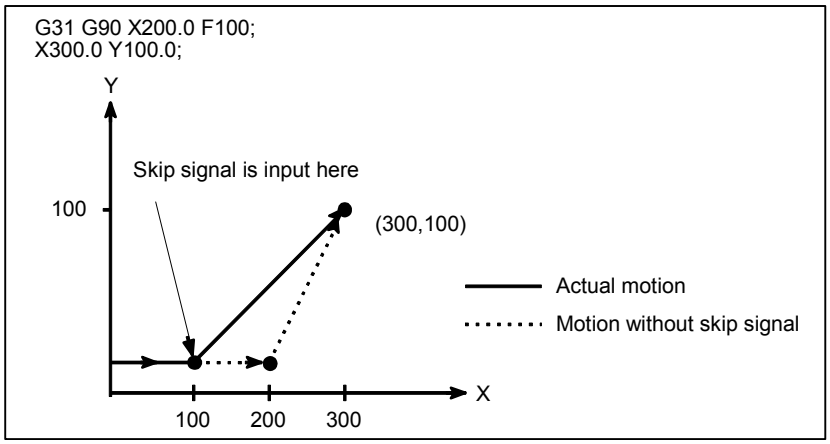


Fig. 4.18 (c) The next block is an absolute programming for 2 axes

4.19 MULTI-STEP SKIP (G31)

In a block specifying P1 to P4 after G31, the multi-step skip function stores coordinates in a custom macro variable when a skip signal (4-point or 8-point ; 8-point when a high-speed skip signal is used) is turned on. In the block where Q1 to Q4 are specified after G04, dwell can be skipped when skip signals (four or eight signals, or eight signals when high-speed skip signals are used) are input.
A skip signal from equipment such as a fixed-dimension size measuring instrument can be used to skip programs being executed.
In plunge grinding, for example, a series of operations from rough machining to spark-out can be performed automatically by applying a skip signal each time rough machining, semi-fine machining, fine-machining, or spark-out operation is completed.

Format

Move command
G31 IP_ F_ P_ ;
IP_ : End point
F_ : Feedrate
P_ : P1 to P4
Dwell
G04X(U,P)_ (Q_);
X(U,P)_ : Dwell time
Q_ : Q1 to Q4

Explanation

Multi-step skip is caused by specifying P1, P2, P3, or P4 in a G31 block. For an explanation of selecting (P1, P2, P3, or P4), refer to the manual supplied by the machine tool builder.
Specifying Q1, Q2, Q3, or Q4 in G04 (dwell command) enables dwell skip in a similar way to specifying G31. A skip may occur even if Q is not specified. For an explanation of selecting (Q1, Q2, Q3, or Q4), refer to the manual supplied by the machine tool builder.

- Correspondence to skip signals

Parameters Nos. 6202 to 6205 can be used to specify whether the 4-point or 8-point skip signal is used (when a high-speed skip signal is used). Specification is not limited to one-to-one correspondence. It is possible to specify that one skip signal correspond to two or more Pn's or Qn's (n=1, 2, 3, 4). Also, bits 0 (DS1) and 7 (DS8) parameter No.6206 can be used to specify dwell.

⚠ CAUTION
Dwell is not skipped when Qn is not specified and bits 0 (DS1) and 7 (DS8) parameter No.6206 are not set.

4.20 HIGH-SPEED SKIP SIGNAL (G31)

The skip function operates based on a high-speed skip signal (connected directly to the NC; not via the PMC) instead of an ordinary skip signal. In this case, up to eight signals can be input.
Delay and error of skip signal input is 0 - 2 msec at the NC side (not considering those at the PMC side).
This high-speed skip signal input function keeps this value to 0.1 msec or less, thus allowing high precision measurement.
For details, refer to the appropriate manual supplied from the machine tool builder.

Format

G31 IP ;

G31; One-shot G code (If is effective only in the block in which it is specified)

4.21 SKIP POSITION MACRO VARIABLE IMPROVEMENT

Overview

In macro variables #100151 to #100200 (#5061 to #5080) for reading the skip position when executing the skip command (G31), by setting bit 7 (SKM) of parameter No. 6007 to 1, it is now possible to read the skip position in the workpiece coordinate system at the time a skip is made even if the workpiece coordinate system setting/selection command is executed after the skip command is executed.

By setting bit 7 (SKM) of parameter No. 6007 to 0, it is possible to read the skip position that reflects the workpiece coordinate system at the time it is read, as usual, if the workpiece coordinate system setting/selection command is executed after the skip command is executed.

Macro variable

The following are the macro variables that can read the position at the time a skip is made even if the workpiece coordinate system setting/selection command is executed after the skip command is executed, by setting bit 7 (SKM) of parameter No. 6007 to 1.

System variable number	System variable name	Attribute	Description
#5061 to #5080	[#_ABSKP[n]]	R	Skip position (workpiece coordinate system) Note) Subscript n represents an axis number (1 to 20).
#100151 to #100200			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).
#5421 to #5440	[#_SKPDTC[n]]	R	Skip position (workpiece coordinate system, detection unit) Note) Subscript n represents an axis number (1 to 20).
#100701 to #100750			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).

4.22 CONTINUOUS HIGH-SPEED SKIP FUNCTION

Overview

The continuous high-speed skip function is used to read absolute coordinates using high-speed skip signals HDI0 to HDI7. Inputting a high-speed skip signal in a G31P90 block causes absolute coordinates to be stored in custom macro variables #5061 to #5080. For a system with more than 20 axes, they are stored in variables #100151 to #100182. An axis movement does not stop even if a signal is input, so that the coordinates of multiple points can be read.

Using bit 5 (CSE) of parameter No. 6201, it is possible to enable both the rising and falling edges of a high-speed skip signal.

For details, refer to the manual supplied by the machine tool builder.

Format

G31 P90 IP ;

G31; One-shot G code (If is effective only in the block in which it is specified)

Explanation

- Custom macro variables

If a high-speed skip signal is input when G31P90 is issued, absolute coordinates are stored in custom macro variables #5061 to #5080. For a system exceeding 20 axes, they are stored in variables #100151 to #100182.

These variables are updated as soon as the axis reaches a new skip position. It is, therefore, necessary to specify a speed so that the axis reaches the next skip position in a period of time longer than it takes for the application to read the variable values.

System with 20 or less axes		System with more than 20 axes	
#5061	Coordinates on the first axis	#100151	Coordinates on the first axis
#5062	Coordinates on the second axis	#100152	Coordinates on the second axis
#5063	Coordinates on the third axis	#100153	Coordinates on the third axis
	:		:
#5079	Coordinates on the 19th axis	#100181	Coordinates on the 31st axis
#5080	Coordinates on the 20th axis	#100182	Coordinates on the 32nd axis

- High-speed skip signal

This function works with a high-speed skip signal only.

Select a high-speed skip signal with bits 0 to 7 (9S1 to 9S8) of parameter No. 6208.

- Ending a block

A G31P90 block is ended when the axis reaches the end point.

Limitation

The continuous high-speed skip function (G31P90) block must be a command for a single axis only. If an attempt is made to specify two or more axes, alarm PS5068 is issued.

4.23 TORQUE LIMIT SKIP

Overview

Executing the move command following G31P99 (or G31P98) while overriding the torque limit[*1] on the servo motor enables cutting feed in to be performed in the same way as in linear interpolation (G01). If, during the movement with this command, the torque of the servo motor reaches the torque limit value (torque limit on the servo motor multiplied by the override) due to pressing or other causes or a skip signal (including a high-speed skip signal) is input, any remaining move commands will be canceled, and the next block is executed. (The operation of canceling any remaining move commands and executing the next block is called a skip operation in the remainder of this document.)

It is possible to override the torque limit on the servo motor with the following command methods:

(1) Execute the torque limit override command in the PMC window.

(2) Execute an address Q command in the same block as that containing a G31P99 (or G31P98) command.

*1 : The torque limit on the servo motor is automatically set to a value conforming to the motor type setting.

Format

G31 P98 Q_ α_ F_

G31 P99 Q_ α_ F_

G31 : Skip command (one-shot G code)
P98 : Performs a skip operation if the torque of the servo motor reaches the limit value.
P99 : Performs a skip operation if the torque of the servo motor reaches the limit value or if a skip signal is input.
Q : Override value for the torque limit
Range of valid settings: 1 to 254 (%)
0 to 255 correspond to 0% to 100%.
The Q command is optional. If omitting it, specify the torque limit command in the PMC window beforehand. If it is omitted, and no torque limit override is set beforehand, alarm PS0035 is issued.
If an attempt is made to specify a value out of range, alarm PS0366 is issued.
The specified override value is effective only to the block in which it is specified.
At the end of the skip operation, the override value returns to the one assumed immediately before the G31 command.
α : Axis address on any one axis
F : Feedrate

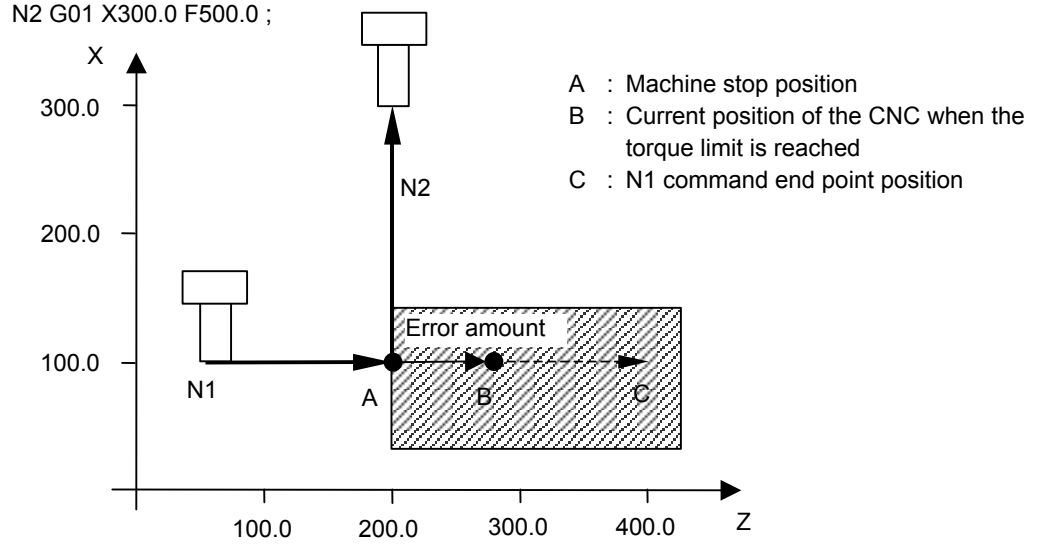
- Conditions for performing a skip operation

Condition	Command	
	G31P98	G31P99
The torque limit value is reached.	A skip operation is performed.	A skip operation is performed.
A skip signal is input.	No skip operation is performed.	A skip operation is performed.

- Operation during a torque limit skip

(Example)

N1 G31 P99 Q50 Z400.0 F100.0 ;
N2 G01 X300.0 F500.0 ;



A : Machine stop position
B : Current position of the CNC when the torque limit is reached
C : N1 command end point position

A torque limit skip presses a specified axis against a previously prepared part or other item while the torque limit command is being executed on the servo motor, and then performs a skip operation when the servo motor reaches the torque limit value. A skip operation is performed with the fact that the torque limit value as detected in the servo motor is reached. It is, therefore, not necessary to input a skip signal using a separate sensor or other device unlike with normal skip functions.

(1) At point A, the machine comes in contact with the object under measurement and stops. At this time, because the torque limit value is not reached, no skip operation is performed, move commands are continuously output, and the current position of the CNC is updated.
(2) Because move commands are output but the machine remains stopped, there occurs a difference (error amount) between the current position of the CNC and the machine position, and torque is applied to the servo motor.
(3) When the torque limit value is reached, a skip operation is performed at machine stop position, point A, and the N2 command is executed. Assuming that the current position of the CNC when the torque limit is reached is point B, the error amount during the torque limit skip is (A - B).

- Torque limit command

If, with the torque limit skip command, no torque limit override value is specified with address Q and no torque limit command is issued from the PMC window, etc., alarm PS0035 is issued.

When no toque limit command is issued, the torque limit override value is either 0% or 100%.

If, in a torque limit skip command block, no torque limit override value is specified with address Q, a torque limit is to be specified as in the program example below.

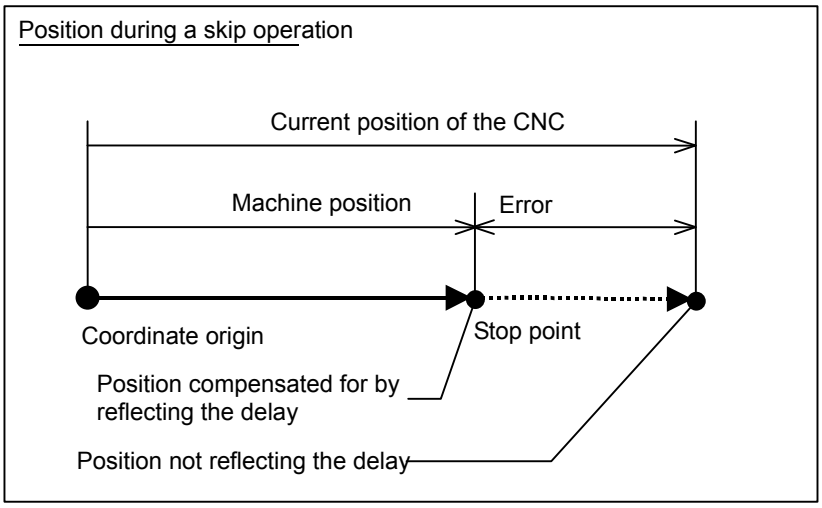
(Program example)
O0012
:
Mxx (Specify a torque limit from the PMC via the window)
:
G31 P99 X200. F100.(Torque limit skip command)
:
G01 X100. F500.(Move command with the torque limit being still effective)
:
Myy (Cancel the torque limit from the PMC)
:
M30

- Positional deviation limit during the torque limit command

While the torque limit skip command is being executed, the positional deviation limit check with the settings of parameters Nos. 1828 and 1829 is not performed. Instead, the positional deviation limit check with the settings of parameter No. 6287 is performed. If the positional deviation exceeds the limit, alarm SV0004 is issued and an instantaneous stop occurs.

- Custom macro variables

When the torque limit skip command is executed, the custom macro system variables #5061 to #5080 (skip signal position; for a system with more than 20 axes, variables #100151 to #100182 are used) store the coordinate position assumed at the end of the skip. In reality, there is a deviation due to the delay of the servo system between the machine position and the current position of the CNC when a skip operation is executed. This deviation can be determined from the positional deviation of the servo. By setting bit 2 (TSE) of parameter No. 6201, it is possible to select whether or not the skip signal position to be stored in system variables should be compensated for with the error (positional deviation) of the servo system.



NOTE
1 Specify only a single axis with the torque limit skip command. If no axis is specified or an attempt is made to specify more than one, alarm PS0369 is issued.
2 Do not issue the torque limit skip command in 3-dimensional coordinate conversion mode or to a parallel axis. Otherwise, alarm PS0035 is issued.
3 Do not issue the torque limit skip command in G41 or G42 mode. Otherwise, alarm PS0035 is issued.
4 A torque limit arrival signal is output regardless of the torque limit skip command.
5 Do not issue the torque limit skip command for an axis being synchronized with synchronization control (such as synchronization control, twin table control, or electronic gear box).
6 Do not specify the torque limit skip command in a continuous block.
7 The higher the movement speed, the larger the error between the position at which the machine stops and the position at which a skip is actually detected. Also, the error increases as the speed is varied during movement. Do not vary the speed with override and so on.

4.24 3-DIMENSIONAL CIRCULAR INTERPOLATION

Overview

Specifying an intermediate and end point on an arc enables circular interpolation in a 3-dimensional space.

Format

The command format is as follows:

G02.4 X_{X1} Y_{Y1} Z_{Z1} $\alpha_{\alpha 1}$ $\beta_{\beta 1}$ **;** First block (mid-point of the arc)

X_{X2} Y_{Y2} Z_{Z2} $\alpha_{\alpha 2}$ $\beta_{\beta 2}$ **;** Second block (end point of the arc)

α,β: Arbitrary axes other than the 3-dimensional circular interpolation axis (up to two axes)
To be specified if necessary.

G03.4 can also be specified instead of G02.4.
G03.4 performs the same operation as G02.4.

Explanation

- G code group

G02.4 and G03.4 are modal G codes of group 01. They therefore remain effective until another G code in group 01 is specified.

- Start point, mid-point, and end point

An arc in a 3-dimensional space is uniquely defined with its start point (current position) and a specified intermediate point and end point, as shown below. Two command blocks are used to define this arc. The first command block specifies the tool path between the start point and intermediate point. The second command block specifies the tool path between the intermediate point and end point.

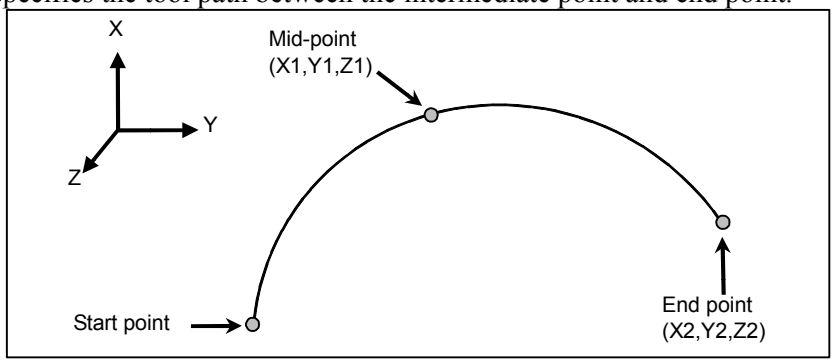


Fig. 4.24 (a) Start, Mid, and End Points

If the modal code is changed by specifying a code such as G01 with the end point not specified, the arc cannot be obtained, and alarm PS5432 is issued. During MDI operation, alarm PS5432 is also issued if a cycle start is applied with only the mid-point specified.

- Movement along axes other than the 3-dimensional circular interpolation axis

In addition to the 3-dimensional circular interpolation axis (X/Y/Z), up to two arbitrary axes (α/β) can be specified at a time. If / are omitted from the first block (mid-point specification) and are specified only in the second block (end point specification), the tool moves to the specified point along the α/β axes during movement from the mid-point of the arc to the end point. If α/β are omitted from the second block (end point specification) and are specified only in the first block (mid-point specification), the tool moves to the specified point along the α/β axes during movement from the start point of the arc to the mid-point.

- Incremental programming

With an incremental programming, the position of the mid-point relative to the start point must be specified in the first block, and the position of the end point relative to the mid-point must be specified in the second block.

- Direction of rotation

The direction of rotation cannot be specified. The movement is the same regardless of whether G02.4 or G03.4 is specified.

- Single block

When operation is performed using a single block, one cycle start causes movement from the start point to the end point. A single-block stop is not performed between the first block (mid-point specification) and the second block (end point specification).

- Start point assumed if 3-dimensional circular interpolations are specified consecutively

If 3-dimensional circular interpolations are specified consecutively, the end point in one interpolation is assumed to be the start point in the next interpolation.

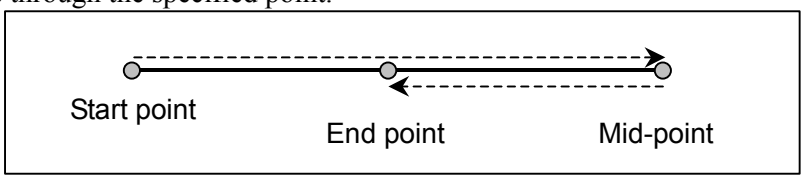
- Velocity commands

As the velocity command, specify the tangential velocity along the arc in the 3-dimensional space.

Limitation

- Cases in which linear interpolation is performed

- f the start point, mid-point, and end-point are on the same line, linear interpolation is performed.
- If the start point coincides with the mid-point, the mid-point coincides with the end point, or the end point coincides with the start point, linear interpolation is performed up to the end point.
- If the start point, mid-point, and end-point are on the same line and the end point lies between the start point and the mid-point, the tool first moves with linear interpolation from the start point to the mid-point, then returns from the mid-point to the end point with linear interpolation. Thus, the tool always passes through the specified point.



- Whole circles

A whole circle (360° arc) cannot be specified. (This corresponds to the case in which linear interpolation is performed, as described earlier.)

- Compensation functions

Before using this function, cancel the compensation functions of group 07, such as cutter radius compensation.

- Manual absolute

While this function is in use, manual intervention is not possible with the manual absolute switch set to the ON position. If intervention is performed, alarm PS5433 is issued when operation restarts.

- Restrictions on commands

In the 3-dimensional circular interpolation mode, the functions listed below can be used, but the states of these functions must not be modified.

- Inch input/metric input (An alarm is issued if a state modification is made with G20 or G21.)
- Mirror image (The state of a signal must not be changed.)
- One-digit F code feed (No feedrate change must be made using the manual handle.)

- Unusable commands

In the 3-dimensional circular interpolation mode, the functions listed below must not be specified. Otherwise, an alarm is issued.

- Exponential interpolation G02.3,G03.3
- Dwell G04
- High-speed machining G05 (Excluding G05P10000 and G05P0)
- AI contour control G05.1Q1,G5.1Q0
- Hypothetical axis interpolation G07
- Cylindrical interpolation G07.1
- Advanced preview control G08 (Use AI high-precision contour control.)
- Polar coordinate interpolation G12.1,G13.1
- Polar coordinate command G15,G16
- Stored stroke check 2 G22
- Reference position return check G27
- Reference position return G28

- 2nd reference position return G30
- 3rd, 4th reference position return G30
- Skip G31
- Threading G33
- Automatic tool length measurement G37
- 3-dimensional cutter compensation G41
- Tool offset G45,G46,G47,G48
- Programmable mirror image G50.1,G51.1
- Local coordinate system G52
- Machine coordinate system G53
- Single direction positioning G60
- Tapping mode G63
- Macro call G65,G66,G67 (Subprogram call is possible.)
- Figure copying G72.1,G72.2
- Canned cycle G73-G79,G80,G81-G89,G98,G99
- Workpiece coordinate system setting G92
- Workpiece coordinate system preset G92.1
- Feed per revolution G95
- Constant surface speed control G96,G97
- NURBS interpolation G06.2
- Workpiece coordinate system setting G54-G59,G54.1
- 3-dimensional coordinate system conversion G68
- Coordinate system rotation G68
- Scaling G50,G51
- Programmable mirror image G50.1,G51.1
- Exact stop mode G61
- Tool radius/tool nose radius compensation G38,G39,G40,G41,G42
- 3-dimensional cutter compensation G40, G41.2, G41.3, G41.4, G41.5, G41.6, G42.2, G42.4, G42.5, G42.6
- Data setting G10
- Exact stop G09
- Auxiliary function
- 2nd auxiliary function
- Spindle function
- Tool function

- Unusable functions

If the following function is specified in the 3-dimensional circular interpolation mode, a warning is output:

- MDI intervention

If any of the following functions is specified in the 3-dimensional circular interpolation mode, the alarm PS5196 is issued:

- Manual operation interrupt
- Tool retract and recover

In the 3-dimensional circular interpolation mode, the following functions cannot be used:

- Sequence number comparison and stop (Stop operation cannot be performed with a sequence number in the 3-dimensional circular interpolation mode.)
- Index table indexing
- Rotary axis control
- Macro executor (Execution macro)
- Manual handle interruption
- Optional chamfering/corner R

- Other limitations

When the following function is used, 3-dimensional circular interpolation cannot be used:

- Arbitrary angular axis control

A limitation may be imposed on other NC command combinations. See the description of each function.

5 FEED FUNCTIONS

Chapter 5, "FEED FUNCTIONS", consists of the following sections:

5.1 OVERVIEW 127
5.2 RAPID TRAVERSE 128
5.3 CUTTING FEED 129
5.4 CUTTING FEEDRATE CONTROL 136
5.5 FEEDRATE INSTRUCTION ON IMAGINARY CIRCLE FOR A ROTARY AXIS 140
5.6 DWELL 144

5.1 OVERVIEW

The feed functions control the feedrate of the tool. The following two feed functions are available:

- Feed functions

1. Rapid traverse
   When the positioning command (G00) is specified, the tool moves at a rapid traverse feedrate set in the CNC (parameter No. 1420).
2. Cutting feed
   The tool moves at a programmed cutting feedrate.

- Override

Override can be applied to a rapid traverse rate or cutting feedrate using the switch on the machine operator's panel.

- Automatic acceleration/deceleration

To prevent a mechanical shock, acceleration/deceleration is automatically applied when the tool starts and ends its movement (Fig. 5.1 (a)).

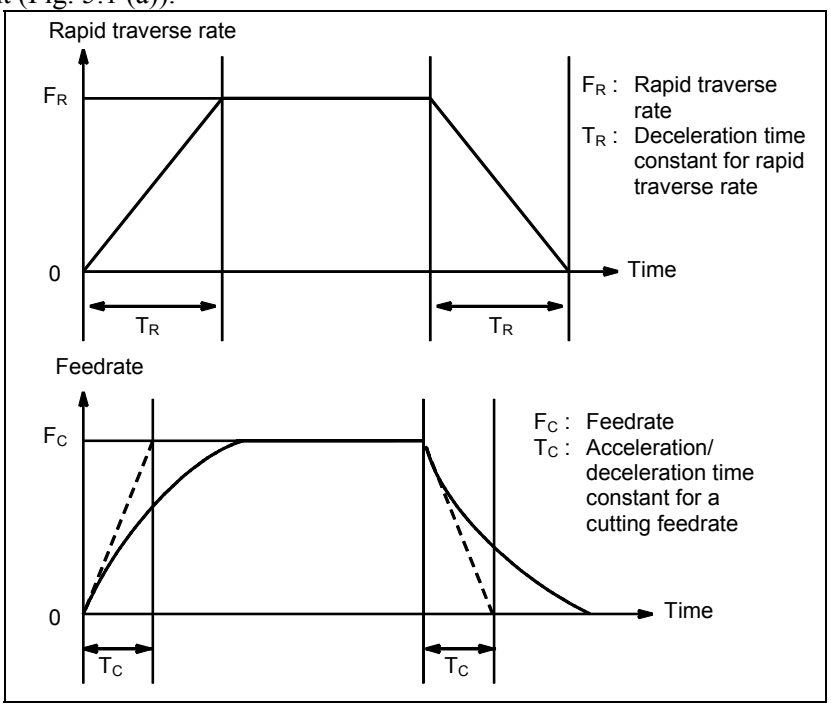


Fig. 5.1 (a) Automatic acceleration/deceleration (example)

- High-speed and high-precious machining function

The machining precision can be improved by using AI contour control function I or II. Refer to the Section of "AI CONTOUR CONTROL FUNCTION I AND AI CONTOUR CONTROL FUNCTION II" in "HIGH-SPEED CUTTING FUNCTION" for detail.

- Tool path in a cutting feed

When the movement direction changes between a specified block and the next block during cutting feed, the tool path may be rounded because of the relationship between the time constant and feedrate (Fig. 5.1 (b)).

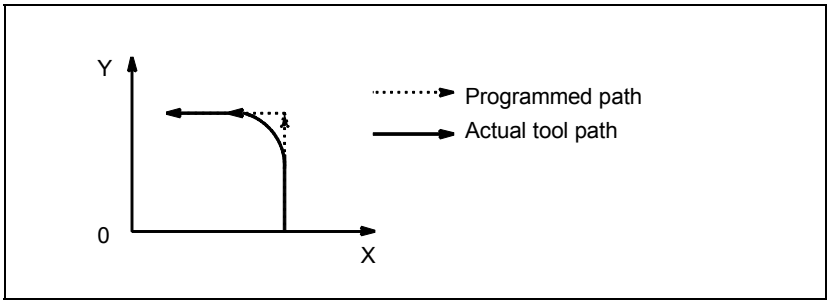


Fig. 5.1 (b) Example of tool path between two blocks

In circular interpolation, a radial error occurs (Fig. 5.1 (c)).

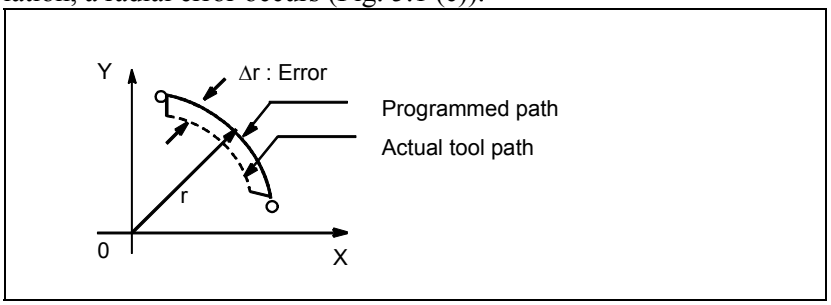


Fig. 5.1 (c) Example of radial error in circular interpolation

The rounded-corner path shown in Fig. 5.1 (b) and the error shown in Fig. 5.1 (c) depend on the feedrate. So, the feedrate needs to be controlled for the tool to move as programmed.

5.2 RAPID TRAVERSE

Format

G00 IP_ ;

G00 : G code (group 01) for positioning (rapid traverse)
IP_ : Dimension word for the end point

Explanation

The positioning command (G00) positions the tool by rapid traverse. In rapid traverse, the next block is executed after the specified feedrate becomes 0 and the servo motor reaches a certain range set by the machine tool builder (in-position check).

A rapid traverse rate is set for each axis by parameter No. 1420, so no rapid traverse feedrate need be programmed.

The following overrides can be applied to a rapid traverse rate with the switch on the machine operator's panel:F0, 25%, 50%, 100%

F0: Allows a fixed feedrate to be set for each axis by parameter No. 1421.

For detailed information, refer to the appropriate manual of the machine tool builder.

5.3 CUTTING FEED

Overview

Feedrate of linear interpolation (G01), circular interpolation (G02, G03), etc. are commanded with numbers after the F code.

In cutting feed, the next block is executed so that the feedrate change from the previous block is minimized.

M

Four modes of specification are available:

1. Feed per minute (G94)
   After F, specify the amount of feed of the tool per minute.
2. Feed per revolution (G95)
   After F, specify the amount of feed of the tool per spindle revolution.
3. Inverse time feed (G93)
   Specify the inverse time (FRN) after F.
4. One-digit F code feed
   Specify a desired one-digit number after F. Then, the feedrate set with the CNC for that number is set.

T

Three modes of specification are available:

1. Feed per minute (G98)
   After F, specify the amount of feed of the tool per minute.
2. Feed per revolution (G99)
   After F, specify the amount of feed of the tool per spindle revolution.
3. Inverse time feed (G93)
   Specify the inverse time (FRN) after F.

Format

M

Feed per minute

G94 ; G code (group 05) for feed per minute

F_ ; Feedrate command (mm/min or inch/min)

Feed per revolution

G95 ; G code (group 05) for feed per revolution

F_ ; Feedrate command (mm/rev or inch/rev)

Inverse time feed (G93)

G93 ; Inverse time feed command G code (05 group)

F_ ; Feedrate command (1/min)

One-digit F code feed

Fn ;

n : Number from 1 to 9

T

Feed per minute
G98 ; G code (group 05) for feed per minute
F_ ; Feedrate command (mm/min or inch/min)
Feed per revolution
G99 ; G code (group 05) for feed per revolution
F_ ; Feedrate command (mm/rev or inch/rev)
Inverse time feed (G93)
G93 ; Inverse time feed command G code (05 group)
F_ ; Feedrate command (1/min)

Explanation

- Direction of the cutting feedrate

Cutting feed is controlled so that the tangential feedrate is always set at a specified feedrate.

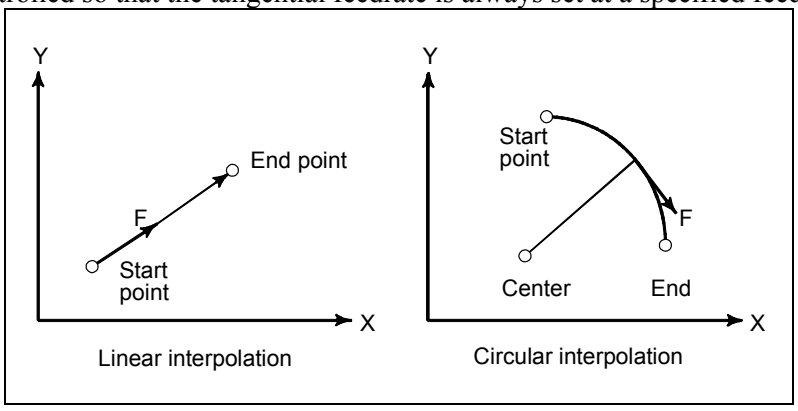


Fig. 5.3 (a) Tangential feedrate (F)

- Feed per minute

After specifying G code for feed per minute (in the feed per minute mode), the amount of feed of the tool per minute is to be directly specified by setting a number after F. G code for feed per minute is a modal code. Once a G code for feed per minute is specified, it is valid until G code for feed per revolution (feed per revolution) is specified.

M

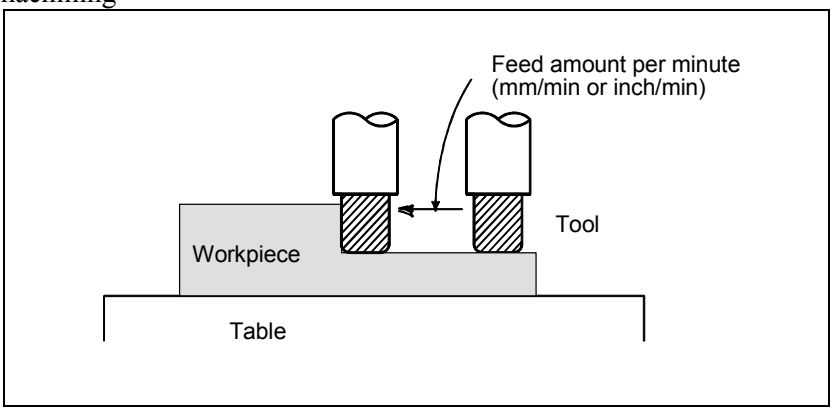
At power-on, the feed per minute mode is set.

T

At power-on, the feed per revolution mode is set.

An override from 0% to 254% (in 1% steps) can be applied to feed per minute with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

- For milling machining



- For lathe cutting

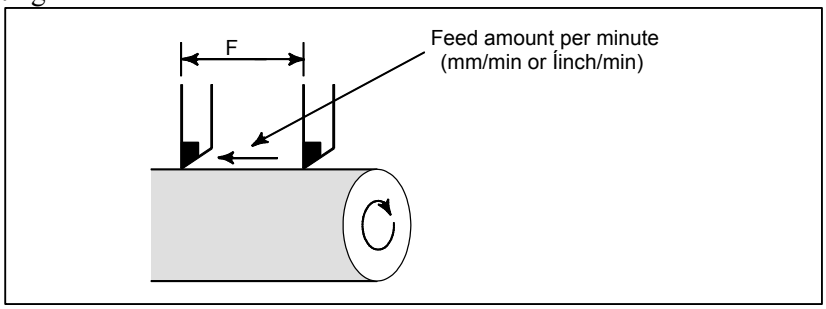


Fig. 5.3 (b) Feed per minute

> ⚠ **CAUTION**
> No override can be used for some commands such as for threading.

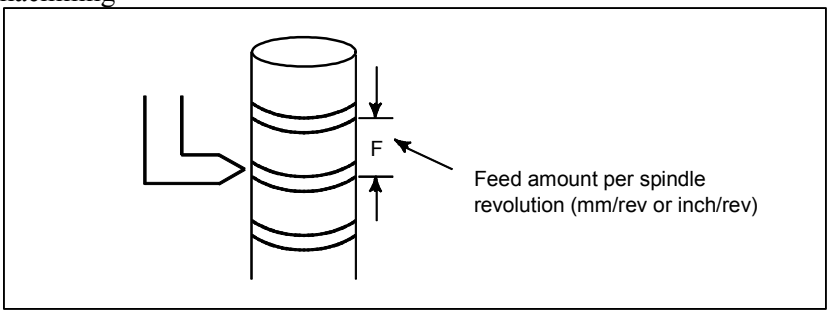
- Feed per revolution

After specifying G code for feed per revolution (in the feed per revolution mode), the amount of feed of the tool per spindle revolution is to be directly specified by setting a number after F. G code for feed per revolution is a modal code. Once a G code for feed per revolution is specified, it is valid until G code for feed per minute (feed per minute) is specified.
An override from 0% to 254% (in 1% steps) can be applied to feed per revolution with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

T

If bit 0 (NPC) of parameter No. 1402 has been set to 1, feed per revolution commands can be specified even when a position coder is not being used. (The CNC converts feed per revolution commands to feed per minute commands.)

- For milling machining



- For lathe cutting

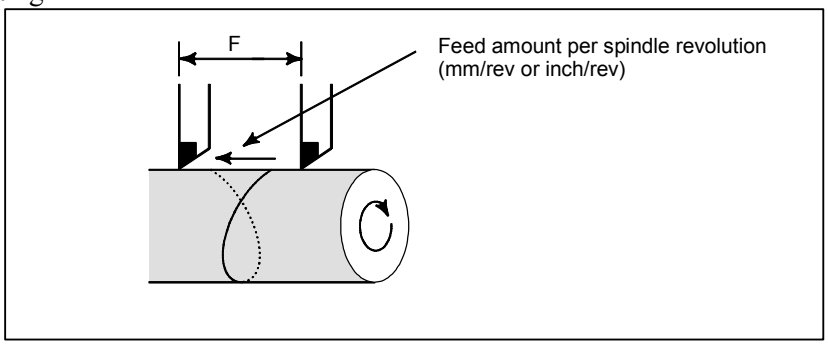


Fig. 5.3 (c) Feed per revolution

> ⚠ **CAUTION**
> When the speed of the spindle is low, feedrate fluctuation may occur.
> The slower the spindle rotates, the more frequently feedrate fluctuation occurs.

- Inverse time feed

When G code for inverse time feed is specified, the inverse time specification mode (G93 mode) is set. Specify the inverse time (FRN) with an F code.
A value from 0.001 to 9999.999 can be specified as FRN, regardless of whether the input mode is inches or metric, or the increment system is IS-B or IS-C.

F code specification value	FRN
F1	0.001
F1 (*1)	1.000
F1.0	1.000
F9999999	9999.999
F9999 (*1)	9999.000
F9999.999	9999.999

> **NOTE**
> *1 Value specified in fixed-point format with bit 0 (DPI) of parameter No. 3401 set to 1

G code for inverse time feed is a modal G code and belongs to group 05 (includes G code for feed per revolution and G code for feed per minute).
When an F value is specified in inverse time specification mode and the feedrate exceeds the maximum cutting feedrate, the feedrate is clamped to the maximum cutting feedrate.
In the case of circular interpolation, the feedrate is calculated not from the actual amount of movement in the block but from the arc radius. This means that actual machining time is longer when the arc radius is longer than the arc distance and shorter when the arc radius is shorter than the arc distance. Inverse time feed can also be used for cutting feed in a canned cycle.

> **NOTE**
> 1 In the inverse time specification mode, an F code is not handled as a modal code and therefore needs to be specified in each block. If an F code is not specified, alarm PS1202, “NO F COMMAND AT G93 ” is issued.
> 2 When F0 is specified in inverse time specification mode, alarm PS0011, “FEED ZERO (COMMAND)” is issued.
> 3 Inverse time feed cannot be used when PMC axis control is in effect.
> 4 If the calculated cutting feedrate is smaller than the allowable range, alarm PS0011, “FEED ZERO (COMMAND)” is issued.

Example

- For linear interpolation (G01)

$$FRN = \frac{1}{time(min)} = \frac{feedrate}{distance}$$

Feedrate: mm/min (for metric input)
inch/min (for inch input)
Distance: mm (for metric input)
inch (for inch input)

- To end a block in 1 (min)

$$FRN = \frac{1}{time(\min)} = \frac{1}{1(\min)} = 1$$ Specify F1.0.

- To end a block in 10 (sec)

$$FRN = \frac{1}{time(\sec)/60} = \frac{1}{10/60(\sec)} = 6$$ Specify F6.0.

- To find the movement time required when F0.5 is specified

$$TIME(\min) = \frac{1}{FRN} = \frac{1}{0.5} = 2$$ 2 (min) is required.

- To find the movement time required when F10.0 is specified

$$TIME(\min) = \frac{1 \times 60}{FRN} = \frac{60}{10} = 6$$ 6 (sec) is required.

- For circular interpolation (G02, G03)

$$FRN = \frac{1}{time(\min)} = \frac{feedrate}{arcradius}$$

Feedrate: mm/min (for metric input)
inch/min (for inch input)
Arc radius: mm (for metric input)
inch (for inch input)

NOTE
In the case of circular interpolation, the feedrate is calculated not from the actual amount of movement in the block but from the arcadias.

- Switching of a modal code in group 05

Usually, if a modal code in group 05 is switched using the feed selection command and the axis command is executed in cutting feed mode without specifying a feedrate (F), the feedrate (F) is inherited as modal data.
If bit 4 (MFC) of parameter No. 13450 is set to 1, the feedrate (F) is cleared if a modal code in group 05 is switched. This causes alarm PS0011, "FEED ZERO (COMMAND)", to be issued in the cutting block. Thus, it is possible to prevent cutting from being executed at an unintended feedrate (F) that is inherited as modal data.

Example

Execute the program below by setting bit 4 (MFC) of parameter No. 13450 to 1.
At the block for switching from G94 to G95 (N06), alarm PS0011 is issued.

```
O0001 ;
N01 G90 G00 X0 Y0 ;
N02 M03 S100 ;
N03 G91 G01 ;
```

```
N04 G94 X10.0 F100.0 ;
N05 Y10.0 ;
N06 G95 X10.0 ; ⇒ Alarm PS0011 is issued.
N07 Y10.0;
M30;
```

NOTE

1 In G93 mode, if the axis command and the feedrate (F) command are not in the same block, alarm PS1202, "NO F COMMAND AT G93" is issued regardless of the setting of this parameter.
2 If the G code of group 05 is cleared due to a reset, by setting bit 6 (CLR) of parameter No. 3402 to 1 and bit 5 (C05) of parameter No. 3406 to 0, so that the modal G code is switched, the feedrate (F) will be cleared even if bit 7 (CFH) of parameter No. 3409 is set to 1.
3 If bit 7 (FC0) of parameter No. 1404 is set to 1, alarm PS0011 is not issued and the block is executed with a feedrate of 0 even if the feed selection command is used to switch the modal code of group 05 and the axis command is executed in cutting feed mode without specifying a feedrate (F). In G93 mode, alarm PS1202 is issued regardless of the setting of the parameter bit FC0.
4 Alarm PS0011 or PS1202 is not used even if the feed selection command is used to switch the modal code of group 05 and the axis command is executed in cutting feed mode without specifying a feedrate (F), provided that the travel distance is 0.
5 Alarms PS0011 and PS1202 are issued if the feed selection command is used to switch the modal code of group 05 and the axis command is executed in cutting feed mode without specifying a feedrate (F), even if cutting feedrate (parameter No. 1411) during automatic operation is set. (M series.)

- One-digit F code feed

M

When a one-digit number from 1 to 9 is specified after F, the feedrate set for that number in a parameters Nos. 1451 to 1459 is used. When F0 is specified, the rapid traverse rate is applied.
The feedrate corresponding to the number currently selected can be increased or decreased by turning on the switch for changing one-digit F feedrate on the machine operator's panel, then by rotating the manual pulse generator.
The increment/decrement, ΔF, in feedrate per scale of the manual pulse generator is as follows:

$$\Delta F = \frac{F\max}{100X}$$

Fmax : Feedrate upper limit for F1-F4 set by parameter No.1460, or feedrate upper limit for F5-F9 set by parameter No.1461
X : Any value of 1-127 set by parameter No.1450

The feedrate set or altered is kept even while the power is off. The current feed rate is displayed on the LCD screen.

- Cutting feedrate clamp

Parameter No. 1430 can be used to specify the maximum cutting feedrate for each axis. When the cutting feedrate along an axis exceeds the maximum feedrate for the axis as a result of interpolation, the cutting feedrate is clamped to the maximum feedrate.

Reference

See Appendix D for range of feedrate command value.

5.4 CUTTING FEEDRATE CONTROL

Cutting feedrate can be controlled, as indicated in Table 5.4 (a).

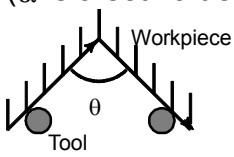
Table 5.4 (a) Cutting Feedrate Control

Function name		G code	Validity of G code	Description
Exact stop		G09	This function is valid for specified blocks only.	The tool is decelerated at the end point of a block, then an in-position check is made. Then the next block is executed.
Exact stop mode		G61	Once specified, this function is valid until G62, G63, or G64 is specified.	The tool is decelerated at the end point of a block, then an in-position check is made. Then the next block is executed.
Cutting mode		G64	Once specified, this function is valid until G61, G62, or G63 is specified.	The tool is not decelerated at the end point of a block, but the next block is executed.
Tapping mode		G63	Once specified, this function is valid until G61, G62, or G64 is specified.	The tool is not decelerated at the end point of a block, but the next block is executed. When G63 is specified, feedrate override and feed hold are invalid.
Automatic corner override	Automatic override for inner corners	G62	Once specified, this function is valid until G61, G63, or G64 is specified.	When the tool moves along an inner corner during tool radius compensation, override is applied to the cutting feedrate to suppress the amount of cutting per unit of time so that a good surface finish can be produced.
	Internal circular cutting feedrate change	-	This function is valid in the tool radius compensation mode, regardless of the G code.	The internal circular cutting feedrate is changed.

NOTE

1 The purpose of in-position check is to check that the servo motor has reached within a specified range (specified with a parameter by the machine tool builder). In-position check is not performed when bit 5 (NCI) of parameter No. 1601 is set to 1.

2 Inner corner angle θ: $2^\circ < \theta \leq \alpha \leq 178^\circ$
(α is a set value)



Format

Exact stop	**G09 IP_ ;**
Exact stop mode	**G61 ;**
Cutting mode	**G64 ;**
Tapping mode	**G63 ;**
Automatic corner override	**G62 ;**

5.4.1 Exact Stop (G09, G61), Cutting Mode (G64), Tapping Mode (G63)

Explanation

The inter-block paths followed by the tool in the exact stop mode, cutting mode, and tapping mode are different (Fig. 5.4.1 (a)).

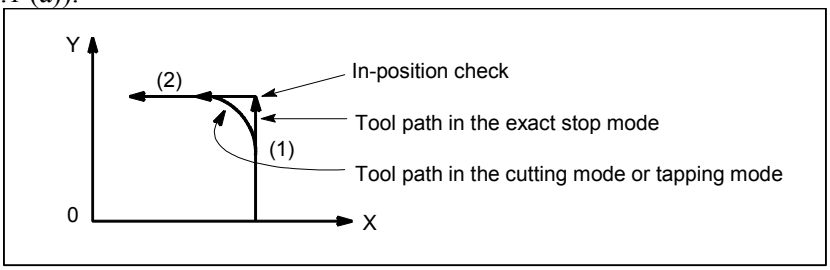


Fig. 5.4.1 (a) Example of tool paths from block (1) to block (2)

> ⚠ **CAUTION**
> The cutting mode (G64 mode) is set at power-on or system clear.

5.4.2 Automatic Corner Override

When tool radius compensation is performed, the movement of the tool is automatically decelerated at an inner corner and internal circular area. This reduces the load on the tool and produces a smoothly machined surface.

5.4.2.1 Automatic override for inner corners (G62)

Explanation

- Override condition

When G62 is specified, and the tool path with tool radius compensation applied forms an inner corner, the feedrate is automatically overridden at both ends of the corner.

There are four types of inner corners (Fig. 5.4.2 (a)).

$2° \leq \theta \leq \theta p \leq 178°$ in Fig. 5.4.2 (a) is a value set with parameter No. 1711. When θ is approximately equal to θp, the inner corner is determined with an error of 0.001° or less.

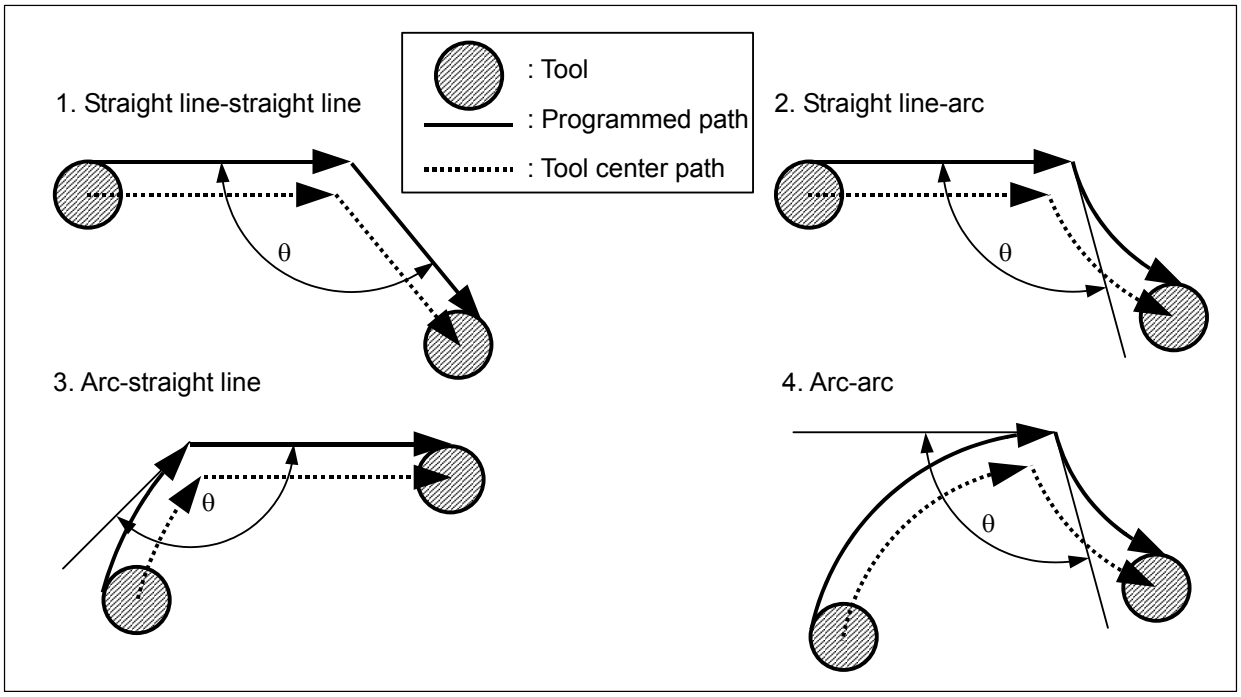


Fig. 5.4.2 (a) Inner corner

- Override range

When a corner is determined to be an inner corner, the feedrate is overridden before and after the inner corner. The distances Ls and Le, where the feedrate is overridden, are distances from points on the tool center path to the corner (Fig. 5.4.2 (b), Fig. 5.4.2 (c), Fig. 5.4.2 (d)). Ls and Le are set with parameters Nos. 1713 and 1714.

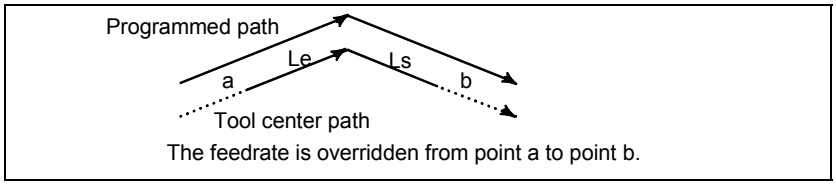


Fig. 5.4.2 (b) Override Range (Straight Line to Straight Line)

When a programmed path consists of two arcs, the feedrate is overridden if the start and end points are in the same quadrant or in adjacent quadrants (Fig. 5.4.2 (c)).

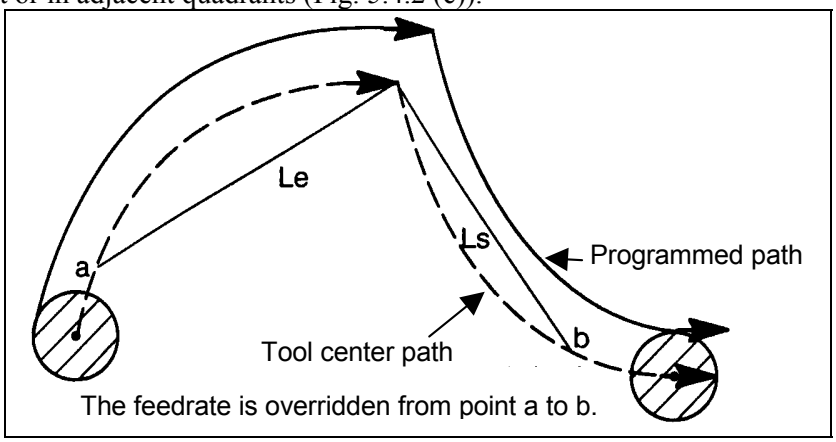


Fig. 5.4.2 (c) Override Range (Arc to Arc)

Regarding program (2) of an arc, the feedrate is overridden from point a to point b and from point c to point d (Fig. 5.4.2 (d)).

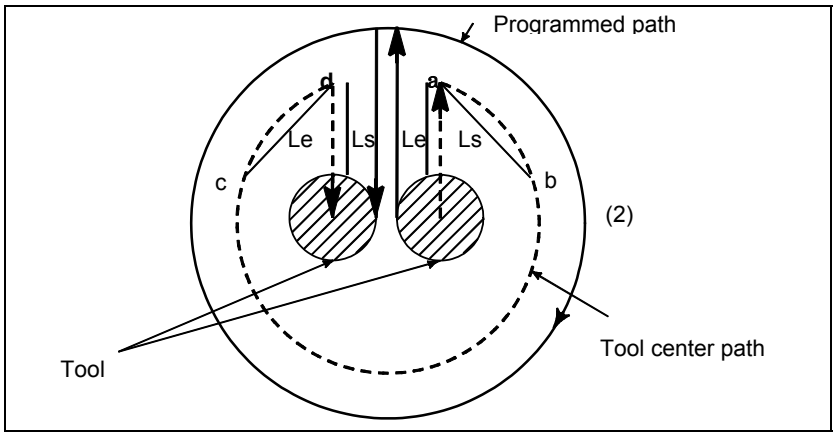


Fig. 5.4.2 (d) Override Range (Straight Line to Arc, Arc to Straight Line)

- Override value

An override value is set with parameter No. 1712. An override value is valid even for dry run and one-digit F code feed specification.
In the feed per minute mode, the actual feedrate is as follows:

$$F = (\text{automatic override for inner corners}) \times (\text{feedrate override})$$

Limitation

- Acceleration/deceleration before interpolation

Override for inner corners is disabled during acceleration/deceleration before interpolation.

- Start-up/G41, G42

Override for inner corners is disabled if the corner is preceded by a start-up block or followed by a block including G41 or G42.

- Offset

Override for inner corners is not performed if the offset is zero.

5.4.2.2 Internal circular cutting feedrate change

For internally offset circular cutting, the feedrate on a programmed path is set to a specified feedrate (F) by specifying the circular cutting feedrate with respect to F, as indicated below (Fig. 5.4.2 (e)). This function is valid in the tool radius compensation mode, regardless of the G62 code.

$$F = \frac{Rc}{Rp}$$

Rc : Tool center path radius
Rp : Programmed radius
It is also valid for the dry run and the one-digit F code feed command.

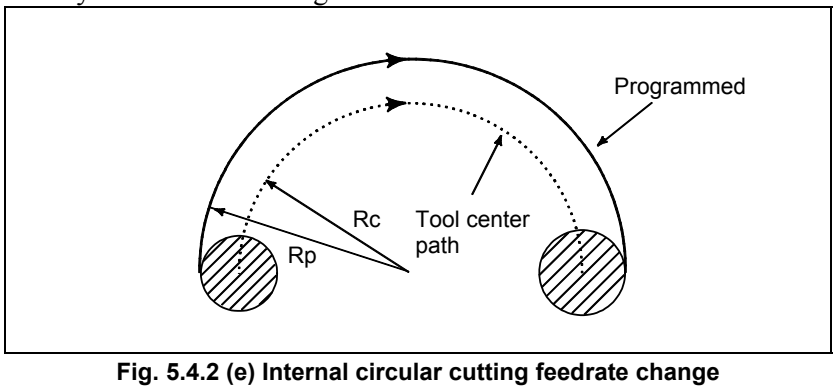


Fig. 5.4.2 (e) Internal circular cutting feedrate change

If Rc is much smaller than Rp, Rc/Rp ≒ 0; the tool stops. A minimum deceleration ratio (MDR) is to be specified with parameter No. 1710. When Rc/Rp≤MDR, the feedrate of the tool is (F×MDR).
If parameter No. 1710 is 0, the minimum deceleration ratio (MDR) is at 100%.

> ⚠ **CAUTION**
> When internal circular cutting must be performed together with override for inner corners, the feedrate of the tool is as follows:
> $F \times \frac{Rc}{Rp} \times (\text{override for the inner corners}) \times (\text{feedrate override})$

5.5 FEEDRATE INSTRUCTION ON IMAGINARY CIRCLE FOR A ROTARY AXIS

Overview

For the feedrate instruction on imaginary circle for a rotary axis, a circle with the radius specified with parameter No. 1465 (which is referred to as an imaginary circle) is considered, and the feedrate of the rotary axis is set to the feedrate on its circumference.

Explanation

- Cutting feedrate

Conventional method

Usually, the feedrate of a rotary axis is set to the rotation feedrate (deg/mm).

Feedrate of liner axis(X axis) $F_X = F \times \frac{\Delta X}{L} \ (mm/min)$

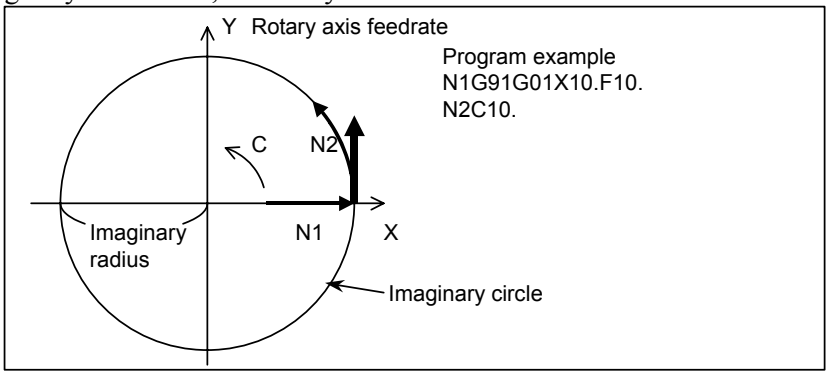
Feedrate of rotary axis(C axis) $F_C = F \times \frac{\Delta C}{L} \ (deg/min)$

Synthetic movement distance $L = \sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2 + \Delta C^2} \ (mm)$

Movement time $T = \frac{L}{F} \ (min)$

Feedrate instruction on imaginary circle of a rotary axis

For the feedrate instruction on imaginary circle for a rotary axis, the feedrate of the rotary axis is set to the movement feedrate on an imaginary circle with the radius specified with parameter No. 1465.
By setting the imaginary radius to 0, the rotary axis can be excluded from feedrate calculation.



Feedrate of liner axis(X axis) $F_X = F \times \frac{\Delta X}{L'} \ (mm/min)$

Feedrate of rotary axis(C axis) $F_C = F \times \frac{\Delta C}{L'} \ (deg/min)$

Synthetic movement distance $L' = \sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2 + \left(\frac{\pi \times l_C \times \Delta C}{180}\right)^2} \ (mm)$

Movement time $T' = \frac{L'}{F} \ (min)$

l_C : imaginary radius(parameter No.1465)

For the feedrate instruction on imaginary circle for a rotary axis, due to the difference in how to determine the movement distance, if a particularly small value is set for the imaginary radius, the movement on the axis will be fast. Pay careful attention to the input to the parameter.

A cutting feedrate clamp is performed based on the per-axis maximum cutting feedrate parameter No. 1430 and the actual speed on the axis (data before conversion). Thus, if a large value is set for the imaginary radius parameter No. 1465, a command can be issued with a feedrate exceeding the setting of the maximum cutting feedrate, and if a small value is set for the imaginary radius, the feedrate is clamped below the setting of the maximum cutting feedrate.
Note that the feedrate instruction on imaginary circle for a rotary axis is also effective to a dry run.

- AI Contour Control

AI contour control is performed on the movement speed on an imaginary circumference. The movement feedrate may, therefore, not be the feedrate for the calculation in the feedrate instruction on imaginary circle for a rotary axis if the feedrate is changed with the feedrate control of AI contour control. The feedrate during AI contour control is clamped with parameter No. 1432. If parameter No. 8465 is not 0, the feedrate is clamped with parameter No. 8465.

- If the imaginary radius is 0 mm

If the imaginary radius is 0 mm, the travel distance on the rotary axis is 0 mm. Thus, the combined travel distance is

$L' = \sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2}$

It is, therefore, possible to exclude the speed component on the rotary axis and regard the movement speed on the linear axis as specified speed F.
If only a rotary axis is specified with this setting, a movement occurs at the maximum cutting feedrate.

Examples

- Example 1

If, in the increment system IS-B,
G91 G01 C10. F10. ;
is specified,

(1) If 10.000 (10 mm) is set for the imaginary radius parameter No. 1465, the calculating formula is:

$$L' = \sqrt{\left(\frac{\pi \times l_C \times \Delta B}{180}\right)^2} = \sqrt{\left(\frac{\pi \times 10_{(mm)} \times 10_{(deg)}}{180}\right)^2} = 1.7453292\cdots_{(mm)}$$

$$F_C = 10_{(mm/\min)} \times \frac{10_{(deg)}}{1.7453292\cdots_{(mm)}} = 57.2957795\cdots_{(deg/min)}$$

$$T' = \frac{L'}{F} = \frac{1.7453292\cdots_{(mm)}}{10_{(mm/min)}} = 0.17453292\cdots_{(min)} = 10.4719755\cdots_{(sec)}$$

Thus, the movement time is about 10.472 sec, and the rotation speed is about 57.296 deg/min. The speed on the imaginary radius of 10.000 mm in Fig. 5.5 (a) above is regarded as the specified speed, 10.000 mm/min.

(2) If 36.000 (36 mm) is set for the imaginary radius parameter No. 1465, the movement time is:

$$L' = \sqrt{\left(\frac{\pi \times l_C \times \Delta B}{180}\right)^2} = \sqrt{\left(\frac{\pi \times 36_{(mm)} \times 10_{(deg)}}{180}\right)^2} = 6.28318530\cdots_{(mm)}$$

$$F_C = 10_{(mm/min)} \times \frac{10_{(deg)}}{6.28318530\cdots_{(mm)}} = 15.9154943\cdots_{(deg/min)}$$

$$T' = \frac{L'}{F} = \frac{6.28318530\cdots_{(mm)}}{10_{(mm/min)}} = 0.628318530\cdots_{(min)} = 37.6991118\cdots_{(sec)}$$

Thus, the movement time is about 37.700 sec, and the rotation speed is about 15.915 deg/min. The speed on the imaginary radius of 36.000 mm in Fig. 5.5 (a) above is regarded as the specified speed, 10.000 mm/min.

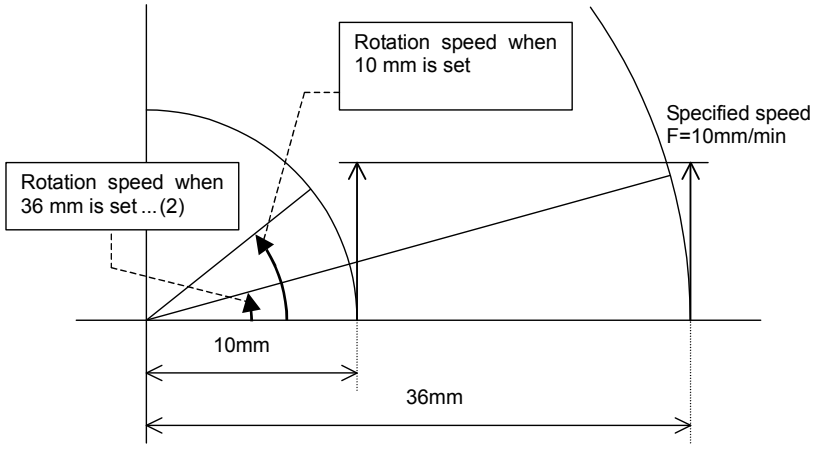


Fig. 5.5 (a)

- Example 2

In a machine in which the tool direction is changed with a rotary axis, such as that shown in Fig. 5.5 (b), it is possible to specify the movement speed at the reference position with F by enabling the feedrate instruction on imaginary circle for a rotary axis and setting the imaginary radius parameter No. 1465 to 0 mm.

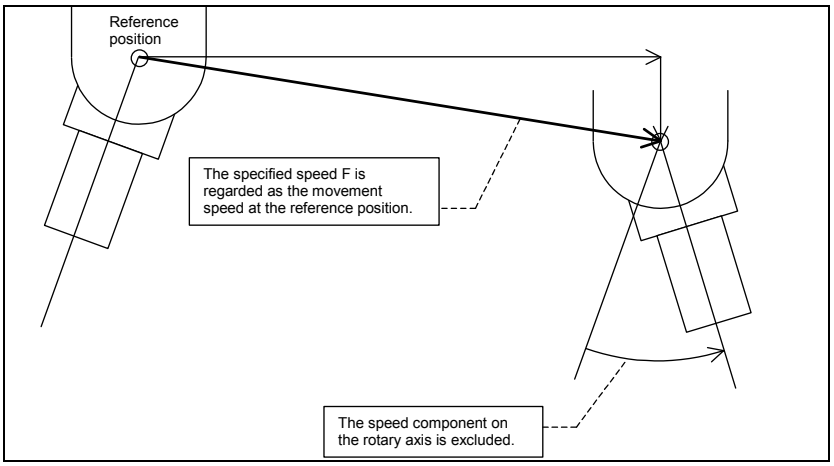


Fig. 5.5 (b)

Limitation

The feedrate instruction on imaginary circle for a rotary axis supports linear interpolation (G01) only. It does not support the following functions:

- Inverse time feed (G93)
- Feed per revolution (G95)
- Normal direction control
- High-speed cycle cutting
- Cylindrical interpolation
- Polar coordinate interpolation
- Tool center point control
- Axis control by PMC
- Manual numerical command

If using parallel axis control or twin table control in the feedrate instruction on imaginary circle for a rotary axis, specify the same value for the imaginary radius parameter No. 1465 for the master and slave axes.

NOTE

1 The feedrate instruction on imaginary circle for a rotary axis is enabled if bit 0 (ROTx) of parameter No. 1006 and bit 0 (RFDx) of parameter No. 1408 are 1.
2 The imaginary radius parameter No. 1465 of the feedrate instruction on imaginary circle for a rotary axis can be overwritten with a programmable parameter input (G10).
3 If the feedrate instruction on imaginary circle for a rotary axis is enabled, setting 0 in the imaginary radius parameter No. 1465 and only a rotary axis is specified, a movement occurs with the maximum cutting feed.
4 Great care is required for the settings of bit 0 (RFDx) of parameter No. 1408 and the imaginary radius parameter No. 1465. Of particular note is that if a small value is set for the imaginary radius and the feedrate instruction on imaginary circle for a rotary axis is used, the movement on the axis is faster than when the command is not used.
5 If feed axis synchronization control is used in the feedrate instruction on imaginary circle for a rotary axis, the settings of parameters Nos. 1408 and 1465 for the master axis are also used for the slave axis.

5.6 DWELL

Format

M

G04 X_; or G04 P_;

X_ : Specify a time or spindle speed (decimal point permitted)
P_ : Specify a time or spindle speed (decimal point not permitted)

T

G04 X_ ; or G04 U_ ; or G04 P_ ;

X_ : Specify a time or spindle speed (decimal point permitted)
U_ : Specify a time or spindle speed (decimal point permitted)
P_ : Specify a time or spindle speed (decimal point not permitted)

Explanation

By specifying a dwell, the execution of the next block is delayed by the specified time. (Dwell per second)
By setting bit 1 (DWL) of parameter No. 3405 in the feed per revolution mode, the execution of the next block is delayed until the rotation count of the spindle reaches the specified number. (Dwell per revolution)

Table 5.6 (a) Command value range of the dwell time (Command by X or U)

Increment system	Command value range	Dwell time unit
IS-A	0.01 to 999999.99	s or rev
IS-B	0.001 to 99999.999	
IS-C	0.0001 to 9999.9999	
IS-D	0.00001 to 999.99999	
IS-E	0.000001 to 99.999999	

Table 5.6 (b) Command value range of the dwell time (Command by P)

Increment system	Command value range	Dwell time unit
IS-A	1 to 99999999	0.01 s or rev
IS-B	1 to 99999999	0.001 s or rev
IS-C	1 to 99999999	0.0001 s or rev
IS-D	1 to 99999999	0.00001 s or rev
IS-E	1 to 99999999	0.000001 s or rev

In the case of dwell per second, the specification unit for dwell time specified with P can be fixed at 0.001 second by setting bit 7 (DWT) of parameter No. 1015 to 1.

NOTE
1 When X, U, or P is specified without a decimal point, the specification unit does not depend on inch/metric input. Depending on whether the X-axis is present, the following increment system is used:
- When the X-axis is present
  The increment system of the X-axis is used.
- When the X-axis is not present
  The increment system of the reference axis is used.

2 When P is specified, bit 7 (IPR) of parameter No. 1004 exercises no influence.

While dwell is being executed, “1” is set for diagnosis display No. 0002.

M

Specify dwell also to make an exact check in the cutting mode (G64 mode).
If the specification of P and X is omitted, an exact stop occurs.

Diagnosis display

2	Dwell execution status

While dwell is being executed, “1” is displayed.

6 REFERENCE POSITION

A CNC machine tool has a special position where, generally, the tool is exchanged or the coordinate system is set, as described later. This position is referred to as a reference position.

Chapter 6, "REFERENCE POSITION", consists of the following sections:

6.1 REFERENCE POSITION RETURN.....146
6.2 FLOATING REFERENCE POSITION RETURN (G30.1) 152

6.1 REFERENCE POSITION RETURN

Overview

- Reference position

The reference position is a fixed position on a machine tool to which the tool can easily be moved by the reference position return function.
For example, the reference position is used as a position at which tools are automatically changed. Up to four reference positions can be specified by setting coordinates in the machine coordinate system in parameters Nos. 1240 to 1243.

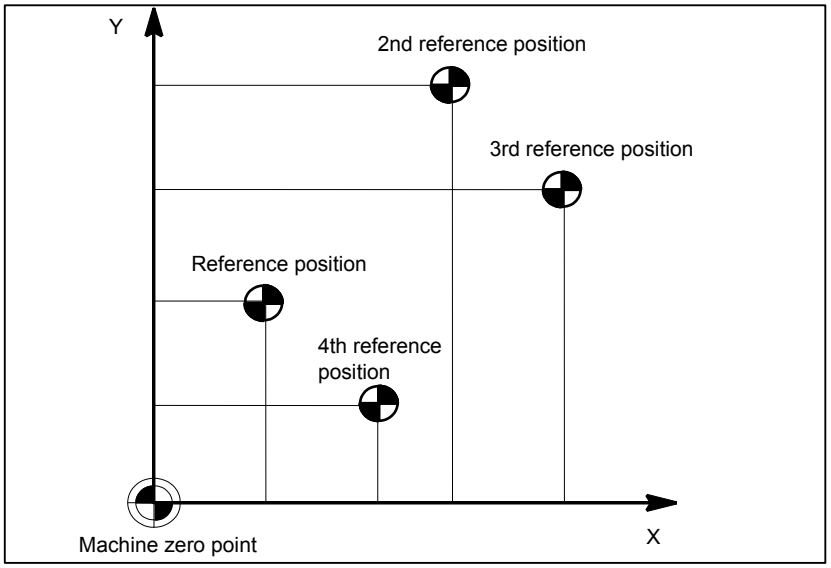


Fig. 6.1 (a) Machine zero point and reference positions

- Automatic reference position return (G28) and movement from the reference position (G29)

The automatic reference position return (G28) function returns tools automatically to the reference position via an intermediate position along a specified axis.
The intermediate position means the relay point for the axes movement in order to avoid collision when there is the interference object on the way of the reference position return.
When reference position return is completed, the lamp for indicating the completion of reference position return goes on. The return from reference position (G29) function moves tools from the reference position to a specified position via an intermediate position along a specified axis.

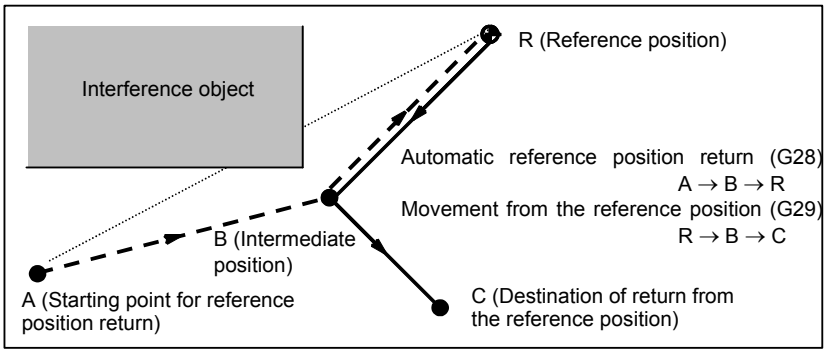


Fig. 6.1 (b) Reference position return and movement from the reference position

If you want to return directly the axes to the reference position from the current position, please specify the incremental command of travel distance zero or the absolute command to current position in the G28 block of the return axes. In this case, an intermediate position is regarded to the current position. The axes are moved to commanded position after passing the starting point of reference position return when the G29 command is specified.

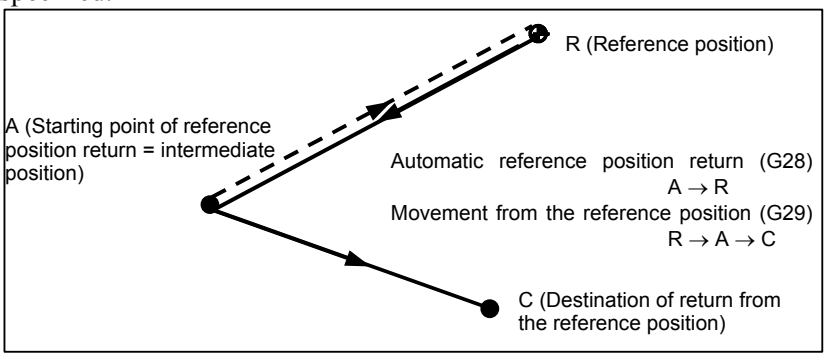


Fig. 6.1 (c) Reference position return and movement from the reference position

- Reference position return check (G27)

The reference position return check (G27) is the function which checks whether the tool has correctly returned to the reference position as specified in the program. If the tool has correctly returned to the reference position along a specified axis, the lamp for the axis for indicating the completion of reference position return goes on.

If the tool has not reached the reference position, an alarm PS0092, "ZERO RETURN CHECK (G27) ERROR" is issued.

When no movement was made along the axis, whether the current position is the reference position is checked.

- In-position check disable reference position return (G28.2, G30.2)

You can disable in-position check at a middle point and reference position by specifying G28.2 or G30.2 as a reference point return command.

Format

- Automatic reference position return and 2nd/3rd/4th reference position return In-position check disable reference position return

G28 IP_;	Reference position return
G30 P2 IP_;	2nd reference position return (P2 can be omitted.)
G30 P3 IP_;	3rd reference position return
G30 P4 IP_;	4th reference position return
G28.2 IP_;	Reference position return
G30.2 P2 IP_;	2nd reference position return (P2 can be omitted.)
G30.2 P3 IP_;	3rd reference position return
G30.2 P4 IP_;	4th reference position return

IP : Specify the intermediate position in the absolute coordinate system. (absolute/incremental programming)
There is no need to calculate an actual travel distance between the intermediate position and the reference position.

- Movement from reference position

G29 IP_;

IP : Specify the destination of return from the reference position in the absolute coordinate system. (absolute/incremental programming)
The intermediate position is determined by G28, G30, or G30.1 specified immediately before this command.

- Reference position return check

G27 IP_;

IP : Specify positioning to the reference position in the absolute coordinate system so as to return to the reference position. (absolute/incremental programming)

Explanation

- Automatic reference position return (G28)

Positioning to the intermediate or reference positions are performed at the rapid traverse rate of each axis. Therefore, for safety, the compensation functions, such as the tool radius compensation and tool length compensation, should be cancelled before executing this command.
The coordinates for the intermediate position are stored in the CNC for the axes for which a value is specified in a G28 block. For the other axes, the previously specified coordinates are used.

(Example)

N1 G28 X40.0 ;	(The tool moves to the reference position along the X-axis and the intermediate position (X40.0) is stored.)
N2 G28 Y60.0 ;	(The tool moves to the reference position along the Y-axis and the intermediate position (Y60.0) is stored.)
N3 G29 X10.0 Y20.0 ;	(The tool moves to the position specified with G29 via the intermediate position (X40.0 Y60.0) previously specified with G28 along the X-axis and Y-axis.)

- 2nd, 3rd, and 4th reference position return (G30)

The 2nd, 3rd, and 4th reference position return (G30) function can be used after the reference positions are established.
The G30 command is generally used when the automatic tool changer (ATC) position differs from the reference position.

- Movement from the reference position (G29)

This function is executed after the tool is returned to the reference position by G28 or G30.
For incremental programming, the command value specifies the incremental value from the intermediate point.
The tool moves to the intermediate and specified positions at the feedrate specified with a parameter.
When the workpiece coordinate system is changed after the tool reaches the reference position through the intermediate point by the G28 command, the intermediate point also shifts to a new coordinate system. If G29 is then commanded, the tool moves to the commanded position through the intermediate point which has been shifted to the new coordinate system.
The same operations are performed also for G30 and G30.1 commands.
After the power is turned on, an alarm PS0305 is issued if an attempt is made to execute G29 (movement from the reference position) before G28 (automatic reference position return), G30 (2nd, 3rd, and 4th reference position return), or G30.1 (floating reference position return) is executed.

- Reference position return check (G27)

G27 command positions the tool at rapid traverse rate. If the tool reaches the reference position, the lamp for indicating the completion of reference position return lights up.
When the tool returns to the reference position along only one axis, the lamp for the axis for indicating the completion of reference position return lights up.
After positioning, if the tool has not reached the reference position along a specified axis, an alarm PS0092, "ZERO RETURN CHECK (G27) ERROR" is issued.
When no movement was made along the axis, whether the current position is the reference position is checked.

- In-position check disable reference position return (G28.2, G30.2)

You can disable in-position check at a middle point and reference position by specifying G28.2 or G30.2 as a reference point return command.
Disabling the in-position check at the middle point and reference position reduces the cycle time.
Note that, if the G28.2 command causes reference position return at low speed, the in-position check is disabled at the middle point but enabled at the reference position.
In the operations of and restrictions on reference position return, G28.2 and G30.2 are the same as G28 and G30, except for in-position check.

- Setting of the reference position return feedrate

Before a coordinate system is established with the first reference position return after power-on, the manual and automatic reference position return feedrates and automatic rapid traverse rate conform to the setting of parameter No. 1428 for each axis.
After a reference position is established upon the completion of reference position return, the manual reference position return feedrate conforms to the setting of the parameter No. 1428 for each axis.

NOTE

1 To this feedrate, a rapid traverse override (F0,25,50,100%) is applied, for which the setting is 100%.
2 After a reference position has been established upon the completion of reference position return, the automatic reference position return feedrate will conform to the ordinary rapid traverse rate.
3 When a value is set for parameter No. 1428, the feedrates conform to the parameter settings shown below.

1420: Rapid traverse rate 1423: Jog feedrate
1424: Manual rapid traverse rate 1428: Reference position return feedrate

	Before a coordinate system is established	After a coordinate system is established
Automatic reference position return (G28)	No. 1428	No.1420
Automatic rapid traverse (G00)	No.1428	No.1420
Manual reference position return (*1)	No.1428	No.1428 (*3)
Manual rapid traverse rate	No.1423 (*2)	No.1424

When parameter No. 1428 is set to 0, the feedrates conform to the parameter settings shown below.

	Before a coordinate system is established	After a coordinate system is established
Automatic reference position return (G28)	No. 1420	No.1420
Automatic rapid traverse (G00)	No.1420	No.1420
Manual reference position return (*1)	No.1424	No.1424 (*3)
Manual rapid traverse rate	No.1423 (*2)	No.1424

*1 By using bit 2 (JZR) of parameter No. 1401, the manual reference position return feedrate can always be set as a jog feedrate.
*2 When bit 0 (RPD) of parameter No. 1401 is 1, the setting of parameter No. 1424 (manual rapid traverse rate) is used.
When the setting of parameter No. 1424 (manual rapid traverse rate) is 0, parameter No. 1420 (rapid traverse rate) is used.
*3 When reference position return without dogs is performed in rapid traverse mode, or when manual reference position return is performed in rapid traverse mode regardless of deceleration dogs after a reference position is established, the reference position return feedrate for each of these functions (setting of bit 1 (DLF) of parameter No. 1404) is used.

Limitation

- Status the machine lock being turned on

The lamp for indicating the completion of reference position return does not go on when the machine lock is turned on, even when the tool has automatically returned to the reference position. In this case, it is not checked whether the tool has returned to the reference position even when a reference position return check command is specified.

- When automatic reference position return (G28) is executed if no reference position is established

When automatic reference position return (G28) is executed if no reference position is established, movement from the intermediate position in a reference position direction is the same as that in manual reference position return.
(This movement is referred to as a low-speed type of automatic reference position return (G28).)

In this case, the tool moves in the direction for reference position return specified in parameter ZMIx (bit 5 of No. 1006). Therefore the specified intermediate position must be a position to which reference position return is possible.

NOTE
When automatic reference position return (G28) is executed after a reference position is established, positioning is performed from the intermediate position to the reference position. This movement is referred to as a high-speed type of automatic reference position return (G28).

- Reference position return check in an offset mode

In an offset mode, the position to be reached by the reference position return check is the position obtained by adding the offset value.
Therefore, if the position with the offset value added is not the reference position, the lamp for indicating the completion of reference position return does not light up, but an alarm is displayed instead. Usually, cancel offsets before G27 is commanded.

- Lighting the lamp when the programmed position does not coincide with the reference position

When the machine tool system is an inch system with metric input, the lamp for indicating the completion of reference position return may also light up even if the programmed position is shifted from the reference position by the least setting increment. This is because the least setting increment of the machine tool system is smaller than its least command increment.

- Automatic reference position return (G28) if roll over is used on a rotation axis (type A)

If, in an absolute command for automatic reference position return (G28) on a rotation axis, the rotation axis is of type A (bit 0 (ROT) of parameter No. 1006 = 1 and bit 1 (ROS) of parameter No. 1006 = 0) and roll over is used (bit 0 (ROA) of parameter No. 1008 = 1), movement to the midpoint follows the setting of bit 1 (RAB) of parameter No. 1008, and movement from the midpoint to the reference position follows the setting of bit 5 (ZMI) of parameter No. 1006.
If, however, bit 6 (RRF) of parameter No. 1008 is set to 1, movement from the midpoint to the reference position also follows the setting of bit 1 (RAB) of parameter No. 1008.

Example

- When the reference position return is executed directly from the current position without passing an intermediate position.

G28G91X0Y0 ; (The axes are directly moved to reference position R from starting point A without passing an intermediate position.)

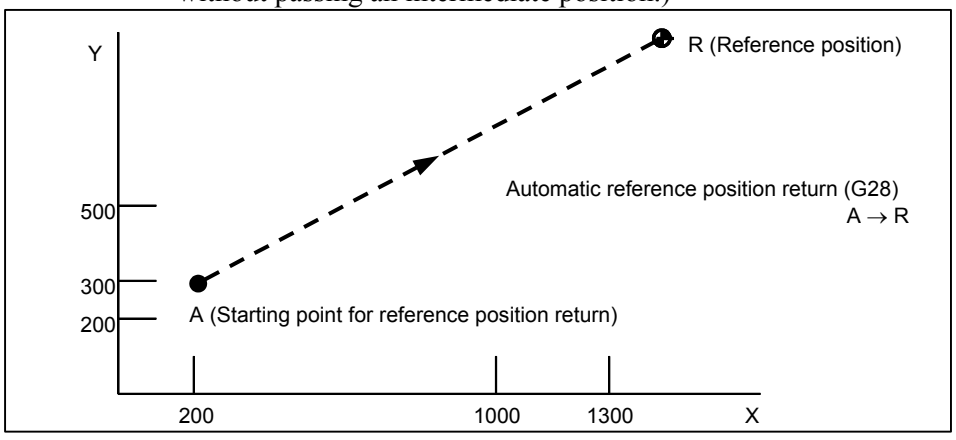


Fig. 6.1 (d) Reference position return and movement from the reference position

- When the movement passes an intermediate position.

G28G90X1000.0Y500.0 ; (Programs movement from A to B. The tool moves to reference position R via intermediate position B.)
T1111 ; (Changing the tool at the reference position)
G29X1300.0Y200.0 ; (Programs movement from B to C. The tool moves from reference position R to C specified with G29 via intermediate position B.)

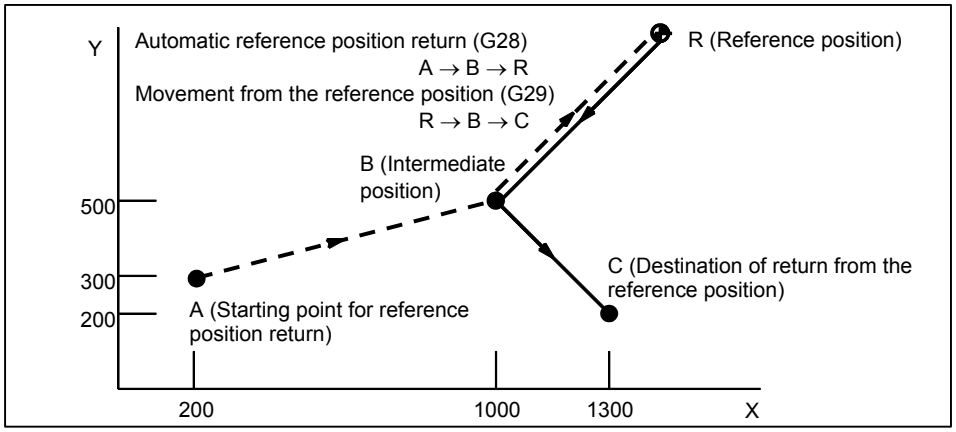


Fig. 6.1 (e) Reference position return and movement from the reference position

6.2 FLOATING REFERENCE POSITION RETURN (G30.1)

Overview

Tools ca be returned to the floating reference position.
A floating reference point is a position on a machine tool, and serves as a reference point for machine tool operation.
A floating reference point need not always be fixed, but can be moved as required.

Format

G30.1 IP_;

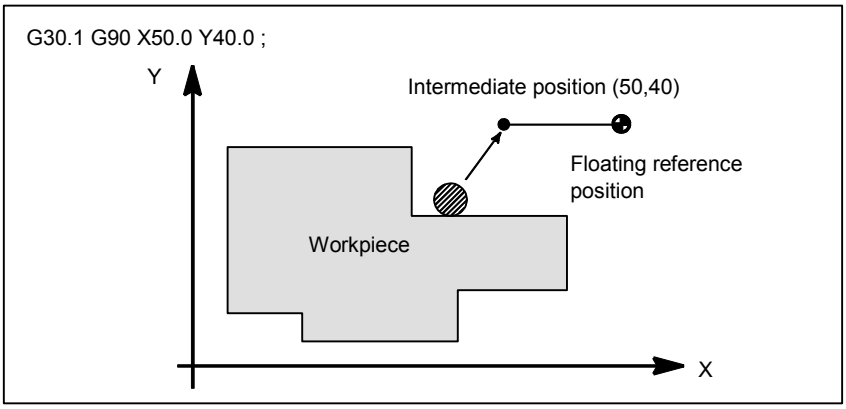
IP_ : Specify the intermediate position to the floating reference position in the absolute coordinate system. (absolute/incremental programming)

Explanation

Generally speaking, on a machining center or milling machine, cutting tools can be replaced only at specific positions. A position where tools can be replaced is defined as the second or third reference point. Using G30 can easily move the cutting tools back to these points.
On some machine tools, the cutting tools can be replaced at any position unless they interfere with the workpiece. With these machines, the cutting tools should be replaced at a position as close to the workpiece as possible so as to minimize the machine cycle time. For this purpose, the tool change position is to be changed, depending on the figure of the workpiece. This operation can easily be performed using this function.
That is, a tool change position suitable for the workpiece is memorized as a floating reference point. Then command G30.1 can easily cause return to the tool change position.
A floating reference point becomes a machine coordinate position memorized by pressing the soft key [SETFRP] on the current positions display screen. The coordinates of the floating reference position are stored with parameter No. 1244.
The G30.1 block first positions the tool at the intermediate point along the specified axes at rapid traverse rate, then further moves the tool from the intermediate point to the floating reference point at rapid

traverse rate. Before using G30.1, cancel the compensation functions, such as tool radius compensation and tool length compensation.
A floating reference point is not lost even if power is turned off.
The function for movement from the reference position (G29) can be specified for moving the tool from the floating reference position.

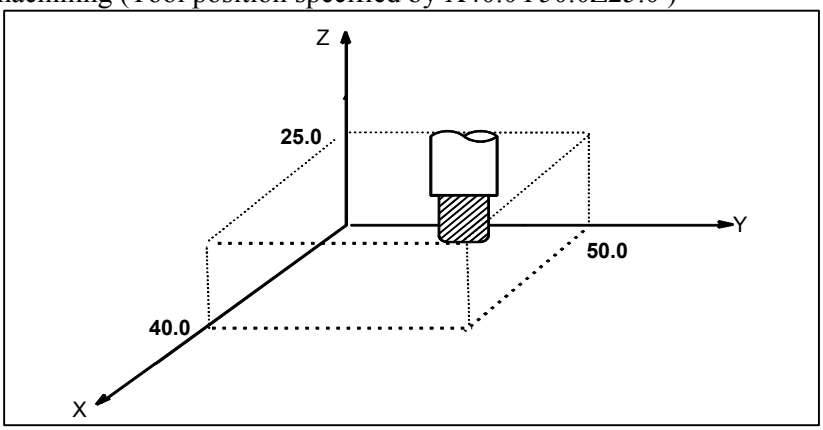
Example



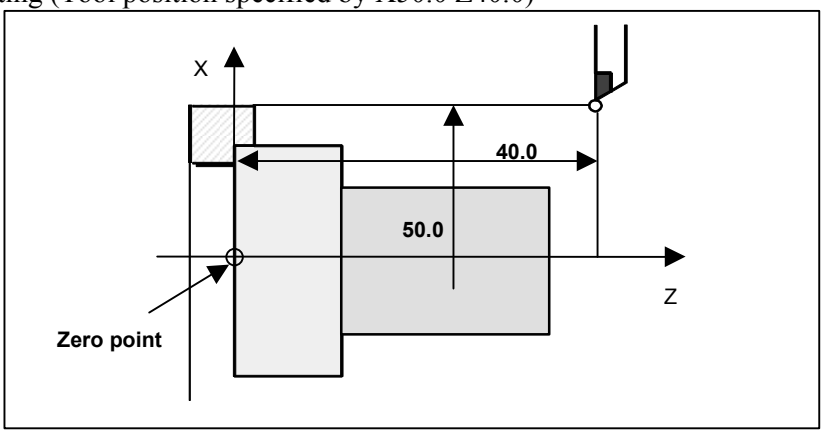
7 COORDINATE SYSTEM

By teaching the CNC a desired tool position, the tool can be moved to the position. Such a tool position is represented by coordinates in a coordinate system. Coordinates are specified using program axes.
When three program axes, the X-axis, Y-axis, and Z-axis, are used, coordinates are specified as follows:
X_Y_Z_
This command is referred to as a dimension word.

- For milling machining (Tool position specified by X40.0Y50.0Z25.0)



- For lathe cutting (Tool position specified by X50.0 Z40.0)



Coordinates are specified in one of following three coordinate systems:
(1) Machine coordinate system
(2) Workpiece coordinate system
(3) Local coordinate system
The number of the axes of a coordinate system varies from one machine to another. So, in this manual, a dimension word is represented as IP_.

7.1 MACHINE COORDINATE SYSTEM

The point that is specific to a machine and serves as the reference of the machine is referred to as the machine zero point. A machine tool builder sets a machine zero point for each machine.
A coordinate system with a machine zero point set as its origin is referred to as a machine coordinate system.
A machine coordinate system is set by performing manual reference position return after power-on (see Section, “MANUAL REFERENCE POSITION RETURN”). A machine coordinate system, once set, remains unchanged until the power is turned off.

The reference position is not always the origin of the machine coordinate system.
(See Item, "Setting a machine coordinate system" described later.)

Format

M

(G90)G53 IP _ P1;

IP_ : Absolute command dimension word
P1 : Enables the high-speed G53 function.

(G90)G53.2 G01 IP _ F _ ;

IP_ : Absolute command dimension word
F_ : Feedrate

T

G53 IP _ P1;

IP_ : Absolute command dimension word
P1 : Enables the high-speed G53 function.

G53.2 G01 IP _ F _ ;

IP_ : Absolute command dimension word
F_ : Feedrate

Explanation

- Selecting a machine coordinate system (G53)

When a command is specified the position on a machine coordinate system, the tool moves to the position by rapid traverse. G53, which is used to select a machine coordinate system, is a one-shot G code; that is, it is valid only in the block in which it is specified on a machine coordinate system. Specify an absolute command for G53. When an incremental command is specified, the G53 command is ignored. When the tool is to be moved to a machine-specific position such as a tool change position, program the movement in a machine coordinate system based on G53.

- High-speed G53 function

This function enables the inter-rapid traverse block overlap function between machine coordinate selection command (G53) and positioning (rapid traverse) command (G00) blocks, thus making it possible to execute the next rapid traverse command (G00) without decelerating to a stop at the end of the machine coordinate selection command (G53). Therefore, high-speed positioning is available even when the machine coordinate selection command (G53) is used.
Specifying P1 in a G53 block enables the high-speed G53 function.

- Selecting a machine coordinate system with feedrate (G53.2)

Positioning of machine coordinate system at a feed rate is available with the command of G53.2. The feed rate can be used in the modal of G01. This function is optional.
G53.2 is one shot G code. Moreover, the tool offset value is temporarily canceled by G53.2 command like G53.
As for the time constant, a usual feed rate are used. At the point of G53.2 command, the axis decelerates to a stop once like G53 command.
Feed per minute, feed per revolution, and inverse time feed of G codes of 05 group are available.
When group 01 G-code except G00/G01 is commanded with G53.2, the alarm PS5372, “IMPROPER MODAL G-CODE” is generated. And, when G53.2 is commanded while group 01 Modal G-code is the one except G00/G01, the alarm PS5372 is generated.

Limitation

- Cancel of the compensation function

When the G53 command is specified, cancel the compensation functions such as the cutter compensation, tool length compensation, tool nose radius compensation, and tool offset beforehand.

- G53 command immediately after power supply turning on

The Reference point by the manual Reference point return or G28 command should return once after turning on the power supply because it should set the machine coordinate system before it instructs in G53.
However, the operation is unnecessary for the machine with the absolute-position detector.

- Specification in the same block

M

Commands G50/G51, G50.1/G51.1, and G68/G69 cannot be specified in the same block where the G53 command is specified.

T

Commands G50/G51 (except for G code system A), G50.1/G51.1, and G68.1/G69.1 cannot be specified in the same block where the G53 command is specified.

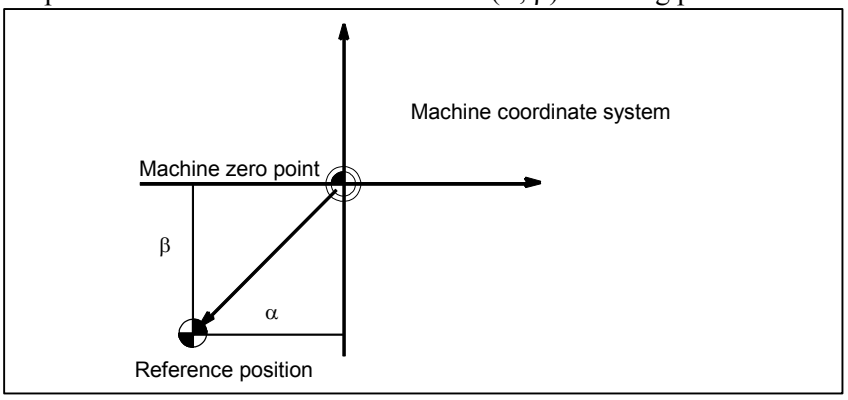
Note

NOTE
G53 is a G code for disabling buffering.

Reference

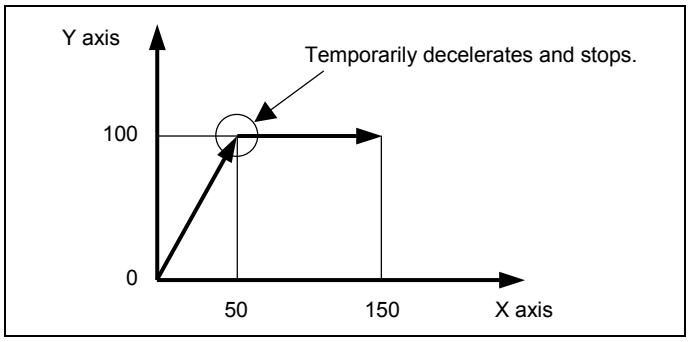
- Setting a machine coordinate system

When manual reference position return is performed after power-on, a machine coordinate system is set so that the reference position is at the coordinate values of (α, β) set using parameter No.1240.



Example (selecting a machine coordinate system with specifying a feedrate)

```
N1 G90 G01 ;
N2 G53.2 X50. Y100. F1000 ;     Absolute command with feedrate F1000
N3 G53.2 X150. F500 ;           Absolute command with feedrate F500
```


Y axis
Temporarily decelerates and stops.
100
0
50
150
X axis

7.2 WORKPIECE COORDINATE SYSTEM

Overview

A coordinate system used for machining a workpiece is referred to as a workpiece coordinate system. A workpiece coordinate system is to be set with the CNC beforehand (setting a workpiece coordinate system).
A machining program sets a workpiece coordinate system (selecting a workpiece coordinate system).
A set workpiece coordinate system can be changed by shifting its origin (changing a workpiece coordinate system).

7.2.1 Setting a Workpiece Coordinate System

A workpiece coordinate system can be set using one of three methods:

(1) Method using a workpiece coordinate system setting G code
A workpiece coordinate system is set by specifying a value in the program after a workpiece coordinate system setting G code.

(2) Automatic setting
If bit 0 (ZPR) of parameter No. 1201 is set to 1, a workpiece coordinate system is automatically set when manual reference position return is performed. (See Section, "MANUAL REFERENCE POSITION RETURN".)
This function is, however, disabled when the workpiece coordinate system option is being used.

(3) Method using a workpiece coordinate system selection G code
Six workpiece coordinate systems can be set beforehand using the MDI unit. Program commands G54 to G59 can be used to select the workpiece axis to be used. (See Subsection, "Displaying and Setting the Workpiece Origin Offset Value".)

When using an absolute command, establish the workpiece coordinate system in any of the above ways.

Format

- Setting a workpiece coordinate system

M

```
(G90) G92 IP_
```

T

```
G50 IP_
```

Explanation

A workpiece coordinate system is set so that a point on the tool, such as the tool tip, is at specified coordinates.

M

If a coordinate system is set using G92 during tool length offset, a coordinate system in which the position before offset matches the position specified in G92 is set. Cutter compensation is cancelled temporarily with G92.

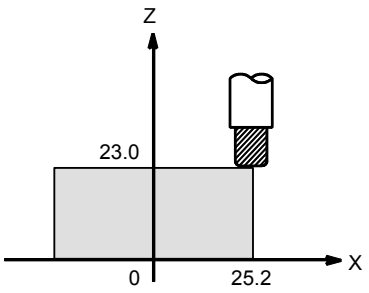
T

If IP_ is an incremental command value, the workpiece coordinate system is defined so that the current tool position coincides with the result of adding the specified incremental value to the coordinates of the previous tool position. If a coordinate system is set using G50 during offset, a coordinate system in which the position before offset matches the position specified in G50 is set.

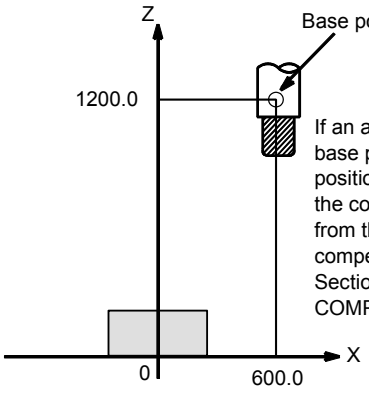
Example

M

(Example 1)
Setting the coordinate system by the G92X25.2Z23.0; command (The tool tip is the start point for the program.)



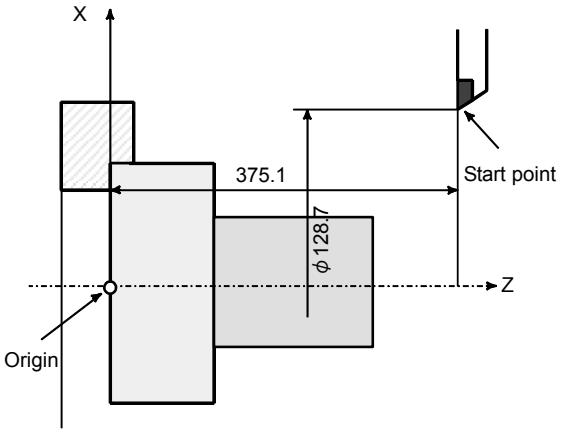
(Example 2)
Setting the coordinate system by the G92X600.0Z1200.0; command (The base point on the tool holder is the start point for the program.)



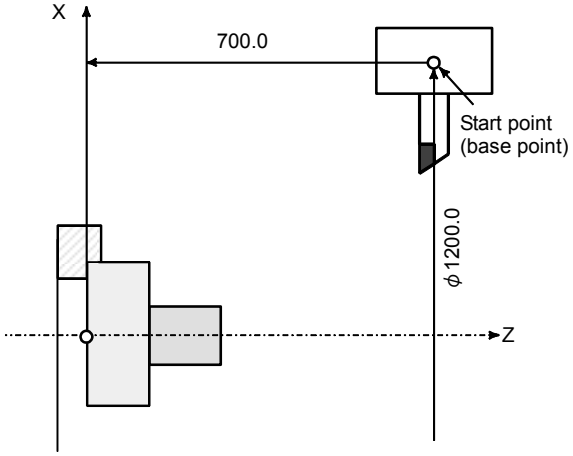
If an absolute command is issued, the base point moves to the commanded position. In order to move the tool tip to the commanded position, the difference from the tool tip to the base point is compensated by tool length offset (see Section, “TOOL LENGTH COMPENSATION (G43, G44, G49)”).

T

(Example 1)
Setting the coordinate system by the G50X128.7Z375.1; command (Diameter designation) (The tool nose is the start point for the program.)



(Example 2)
Setting the coordinate system by the G50X1200.0Z700.0; command (Diameter designation) (The base point on the turret is the start point for the program.)



> **⚠ CAUTION**
> The set workpiece coordinate system depends on diameter programming or radius programming.

Notes

- Command for setting a workpiece coordinate system in the tool length compensation mode

Executing a workpiece coordinate system setting G code command (G92 or, for G code system A in the lathe system, G50) presets a coordinate system in such a way that the specified position will be a pre-compensation position.
However, this G code cannot be used together with a block where tool length compensation vectors vary. For example, it cannot be used together with the following blocks.

Example
1 Block in which G43/G44 is issued
2 Block which is in the G43 or G44 mode and in which an H code is issued
3 Block which is in the G43 or G44 mode and in which G49 is issued
4 Block in which, in the G43 or G44 mode, compensation vectors are canceled using a G code such as G28 or G53 and then resumed again

When presetting a workpiece coordinate system, using the workpiece coordinate system setting G code, do not stop in the previous block to change a tool length compensation offset selected, for example, with the MDI.

7.2.2 Selecting a Workpiece Coordinate System

The user can choose from set workpiece coordinate systems as described below. (For information about the methods of setting, see Subsection, "Setting a Workpiece Coordinate System".)

(1) Once a workpiece coordinate system is set by a workpiece coordinate system setting G code or by automatic workpiece coordinate system setting, absolute commands indicate positions in the workpiece coordinate system.

(2) Choosing from six workpiece coordinate systems set using the MDI unit
By specifying a G code from G54 to G59, one of the workpiece coordinate systems 1 to 6 can be selected.
G54 : Workpiece coordinate system 1 G55 : Workpiece coordinate system 2
G56 : Workpiece coordinate system 3 G57 : Workpiece coordinate system 4
G58 : Workpiece coordinate system 5 G59 : Workpiece coordinate system 6
Workpiece coordinate system 1 to 6 are established after reference position return after the power is turned on. When the power is turned on, G54 coordinate system is selected.

When bit 2 (G92) of parameter No. 1202 is set to 1, executing the workpiece coordinate system setting G92 code command results in the issue of an alarm PS0010. This is designed to prevent the user from confusing coordinate systems.

⚠ CAUTION
The set workpiece origin offset value depends on diameter programming or radius programming.

Example

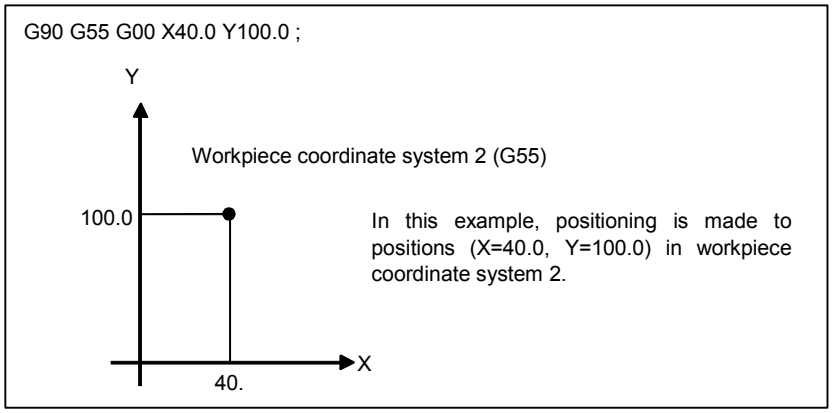


Fig. 7.2.2 (a)

7.2.3 Changing Workpiece Coordinate System

The six workpiece coordinate systems specified with G54 to G59 can be changed by changing an external workpiece origin offset value or workpiece origin offset value.
Three methods are available to change an external workpiece origin offset value or workpiece origin offset value.

(1) Inputting from the MDI unit (see Subsection, “Displaying and Setting the Workpiece Origin Offset Value”.)
(2) Programming (using a programmable data input G code or a workpiece coordinate system setting G code)
(3) Using the external data input function
An external workpiece origin offset value can be changed by input signal to CNC. Refer to machine tool builder's manual for details.

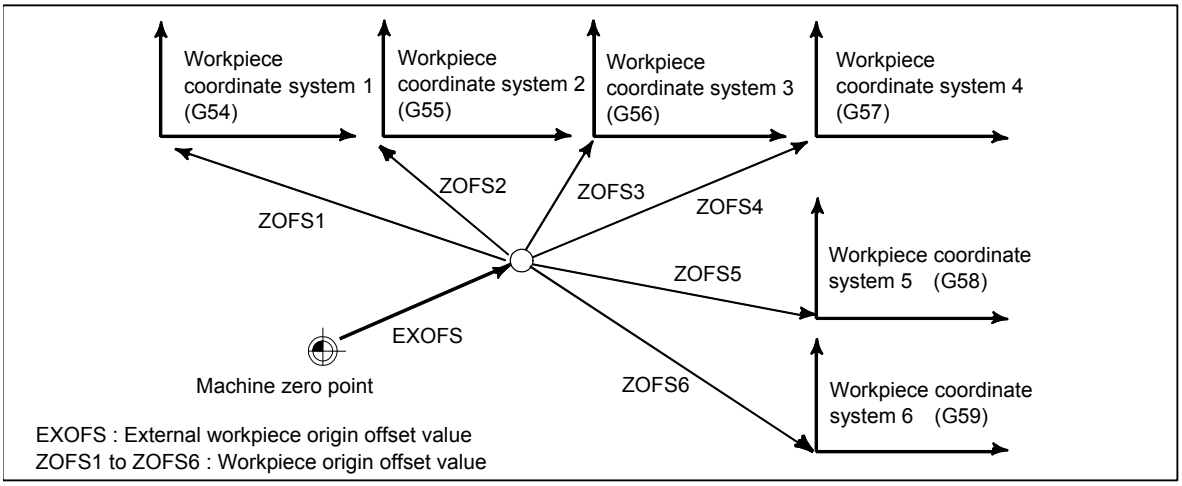


Fig. 7.2.3 (a) Changing an external workpiece origin offset value or workpiece origin offset value

Format

- Changing by inputting programmable data

G10 L2 Pp IP_;

p=0 : External workpiece origin offset value
p=1 to 6 : Workpiece origin offset value correspond to workpiece coordinate system 1 to 6
IP_ : For an absolute command, workpiece origin offset for each axis.
For an incremental command, value to be added to the set workpiece origin offset for each axis (the result of addition becomes the new workpiece origin offset).

- Changing by setting a workpiece coordinate system

M

G92 IP_ ;

T

G50 IP_ ;

Explanation

- Changing by inputting programmable data

By specifying a programmable data input G code, the workpiece origin offset value can be changed for each workpiece coordinate system.

- Changing by setting a workpiece coordinate system

By specifying a workpiece coordinate system setting G code, the workpiece coordinate system (selected with a code from G54 to G59) is shifted to set a new workpiece coordinate system so that the current tool position matches the specified coordinates (IP_).
Then, the amount of coordinate system shift is added to all the workpiece origin offset values. This means that all the workpiece coordinate systems are shifted by the same amount.

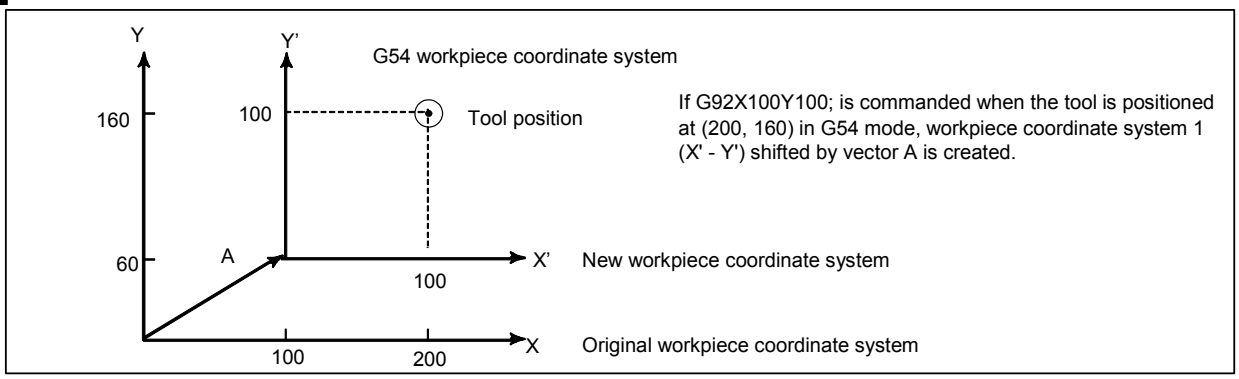
> **⚠ CAUTION**
> When a coordinate system is set with workpiece coordinate system setting G92 code command after an external workpiece origin offset value is set, the coordinate system is not affected by the external workpiece origin offset value. When G92X100.0Z80.0; is specified, for example, the coordinate system having its current tool reference position at X = 100.0 and Z = 80.0 is set.

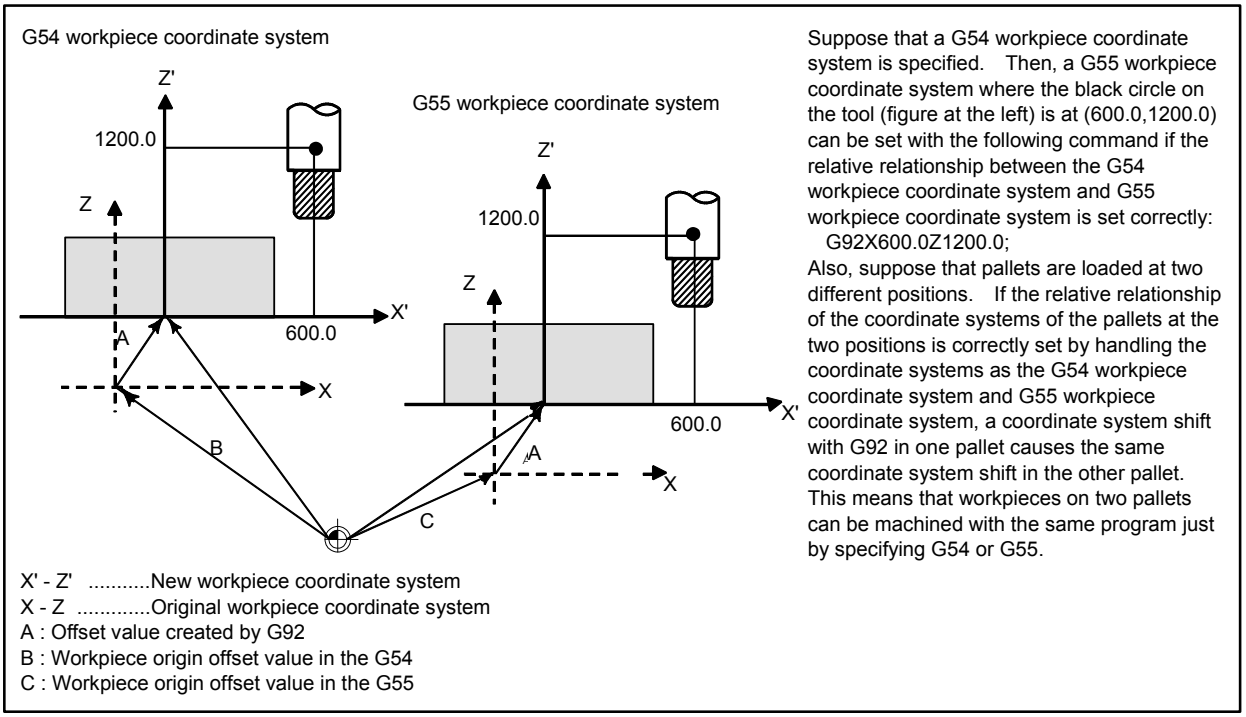
T

If IP is an incremental command value, the workpiece coordinate system is defined so that the current tool position coincides with the result of adding the specified incremental value to the coordinates of the previous tool position. (Coordinate system shift)

Example

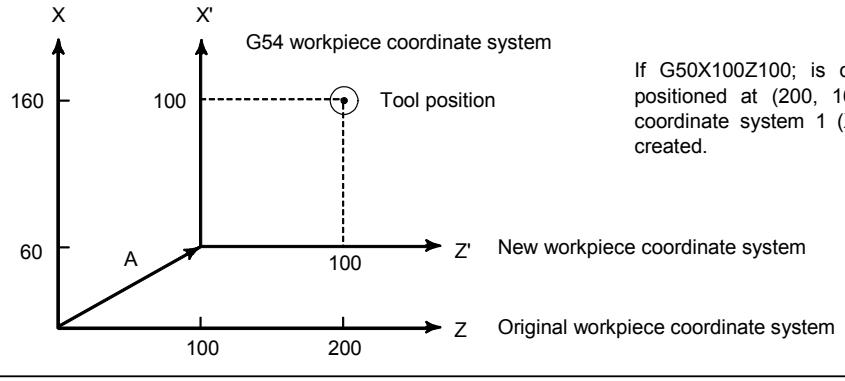
M



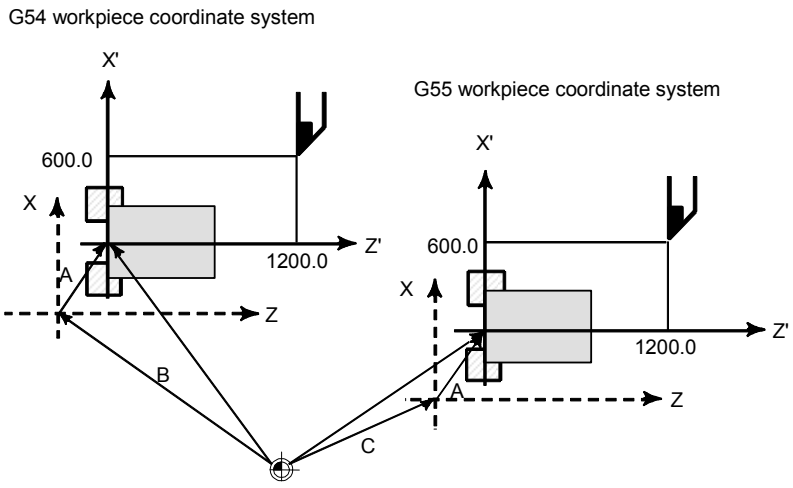


Example

T



If G50X100Z100; is commanded when the tool is positioned at (200, 160) in G54 mode, workpiece coordinate system 1 (X' - Z') shifted by vector A is created.



X' - Z' ..... New workpiece coordinate system
X - Z ..... Original workpiece coordinate system
A : Offset value created by G50
B : Workpiece origin offset value in G54
C : Workpiece origin offset value in G55

Suppose that a G54 workpiece coordinate system is specified. Then, a G55 workpiece coordinate system where the black tool nose point on the tool (figure at the left) is at (600.0,1200.0) can be set with the following command if the relative relationship between the G54 workpiece coordinate system and G55 workpiece coordinate system is set correctly:

G50X600.0Z1200.0;

Also, suppose that loading is performed at two different places and that the G54 and G55 workpiece coordinate systems are set for these two places. If the relative relationship between the coordinate systems in the two places is correctly set, a coordinate system shift with G50 in one loading place causes the same coordinate system shift in the other loading place. This means that workpieces on the two loading positions can be machined with the same program just by specifying G54 or G55.

7.2.4 Workpiece Coordinate System Preset (G92.1)

The workpiece coordinate system preset function presets a workpiece coordinate system shifted by manual intervention to the pre-shift workpiece coordinate system. The latter system is displaced from the machine zero point by a workpiece origin offset value.

There are two methods for using the workpiece coordinate system preset function. One method uses a programmed command. The other uses MDI operations on the absolute position display screen, relative position display screen, and overall position display screen (see Subsection, “Workpiece Coordinate System Preset”).

Format

M

G92.1 IP 0 ;

IP 0 : Specifies axis addresses subject to the workpiece coordinate system preset operation. Axes that are not specified are not subject to the preset operation.

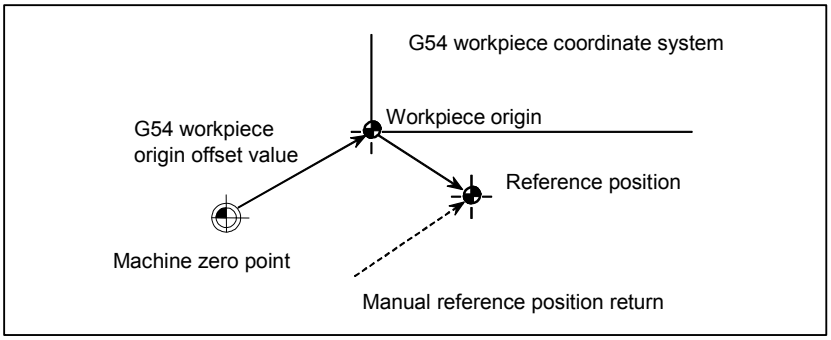
T

G50.3 IP0 ; (G92.1 IP 0; for G code system B or C)

IP 0 : Specifies axis addresses subject to the workpiece coordinate system preset operation. Axes that are not specified are not subject to the preset operation.

Explanation

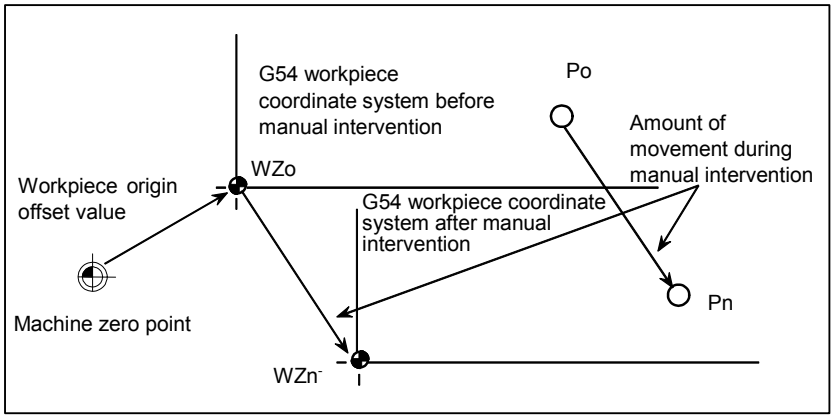
When manual reference position return operation is performed in the reset state, a workpiece coordinate system is shifted by the workpiece origin offset value from the machine coordinate system zero point. Suppose that the manual reference position return operation is performed when a workpiece coordinate system is selected with G54. In this case, a workpiece coordinate system is automatically set which has its origin displaced from the machine zero point by the G54 workpiece origin offset value; the distance from the origin of the workpiece coordinate system to the reference position represents the current position in the workpiece coordinate system.



If an absolute position detector is provided, the workpiece coordinate system automatically set at power-up has its origin displaced from the machine zero point by the G54 workpiece origin offset value. The machine position at the time of power-up is read from the absolute position detector and the current position in the workpiece coordinate system is set by subtracting the G54 workpiece origin offset value from this machine position. The workpiece coordinate system set by these operations is shifted from the machine coordinate system using the commands and operations listed below.

(a) Manual intervention performed when the manual absolute signal is off
(b) Move command executed in the machine lock state
(c) Movement by handle interruption
(d) Operation using the mirror image function
(e) Shifting the workpiece coordinate system by setting the local coordinate system or workpiece coordinate system

In the case of (a) above, the workpiece coordinate system is shifted by the amount of movement during manual intervention.



In the operation above, a workpiece coordinate system once shifted can be preset using G code (G92.1) specification or MDI operation to a workpiece coordinate system displaced by a workpiece origin offset value from the machine zero point.
Bit 3 (PPD) of parameter No. 3104 specifies whether to preset relative coordinates (RELATIVE) as well as absolute coordinates. Moreover, the difference below the least input increment between absolute coordinate point and relative coordinate point is corrected to the value of the preset(or origin) of relative coordinates when bit 4 (RPP) of parameter No.3129 is set to 1.
When no workpiece coordinate system option (G54 to G59) is selected, the workpiece coordinate system is preset to the coordinate system with its origin placed at the reference position.

Limitation

- Tool radius · tool nose radius compensation, tool length compensation, tool offset

When using the workpiece coordinate system preset function, cancel compensation modes: Tool radius · tool nose radius compensation, tool length compensation, and tool offset. If the function is executed without canceling these modes, compensation vectors are cancelled.

M

- Tool length compensation

When using the workpiece coordinate system preset function, cancel tool length compensation. If the function is executed without canceling these modes, compensation vectors are cancelled.

- Program restart

The workpiece coordinate system preset function is not executed during program restart.

- Prohibited modes

Do not use the workpiece coordinate system preset function when the scaling, coordinate system rotation, programmable image, or figure copy mode is set.

7.2.5 Addition of Workpiece Coordinate System Pair (G54.1 or G54)

Besides the six workpiece coordinate systems (standard workpiece coordinate systems) selectable with G54 to G59, 48 or 300 additional workpiece coordinate systems (additional workpiece coordinate systems) can be used.

Format

- Selecting the additional workpiece coordinate systems

G54.1Pn ; or **G54Pn ;**

Pn : Codes specifying the additional workpiece coordinate systems
n : 1 to 48 or 1 to 300

- Setting the workpiece origin offset value in the additional workpiece coordinate systems (G10)

G10L20Pn IP_;

Pn : Codes specifying the workpiece coordinate system for setting the workpiece origin offset value
n : 1 to 48 or 1 to 300

IP_ : Axis addresses and a value set as the workpiece origin offset

Explanation

- Selecting the additional workpiece coordinate systems

When a P code is specified together with G54.1 (G54), the corresponding coordinate system is selected from the additional workpiece coordinate systems (1 to 48 or 1 to 300).

A workpiece coordinate system, once selected, is valid until another workpiece coordinate system is selected. Standard workpiece coordinate system 1 (selectable with G54) is selected at power-on.

G54.1 P1..... Additional workpiece coordinate system 1
G54.1 P2..... Additional workpiece coordinate system 2
:
G54.1 P48..... Additional workpiece coordinate system 48
:
G54.1 P300..... Additional workpiece coordinate system 300

As with the standard workpiece coordinate systems, the following operations can be performed for a workpiece origin offset in an additional workpiece coordinate system:

(1) The workpiece origin offset value setting screen can be used to display and set a workpiece origin offset value.
(2) The G10 function enables a workpiece origin offset value to be set by programming (see Subsection, “Changing Workpiece Coordinate System”).
(3) A custom macro allows a workpiece origin offset value to be handled as a system variable.
(4) Workpiece origin offset data can be entered or output as external data.
(5) The PMC window function enables workpiece origin offset data to be read as program command modal data.

- Setting the workpiece origin offset value in the additional coordinate systems (G10)

When a workpiece origin offset value is specified using an absolute value, the specified value is the new offset value. When it is specified using an incremental value, the specified value is added to the current offset value to obtain a new offset value.

Limitation

- Specifying P codes

A P code must be specified after G54.1 (G54). If G54.1 is not followed by a P code in the same block, additional workpiece coordinate system 1 (G54.1P1) is assumed.

If a value not within the specifiable range is specified in a P code, an alarm PS0030 is issued.

P codes other than workpiece offset numbers cannot be specified in a G54.1 (G54) block.

Example 1) G54.1G04P1000;
Example 2) G54.1M98P48;

7.2.6 Automatic Coordinate System Setting

When bit 0 (ZPR) of parameter No. 1201 for automatic coordinate system setting is 1, a coordinate system is automatically determined when manual reference position return is performed.
Once α, β, and γ are set with parameter No. 1250, a workpiece coordinate system is set upon reference position return so that the base point on the tool holder or the tip of the basic tool is positioned at X = α, Y = β, and Z = γ.
This processing occurs as if the following are specified at the reference position:

M

G92XαYβZγ;

T

G50XαZγ;

This function cannot be used, however, when the workpiece coordinate system option is selected.

T

When the setting of a workpiece coordinate system shift amount is other than 0, a workpiece coordinate system shifted by the amount is set.

7.2.7 Workpiece Coordinate System Shift

T

Explanation

When the coordinate system actually set by the G50 command or the automatic system setting deviates from the programmed workpiece system, the set coordinate system can be shifted (see Section, "MANUAL REFERENCE POSITION RETURN").
Set the desired shift amount in the workpiece coordinate system shift memory.

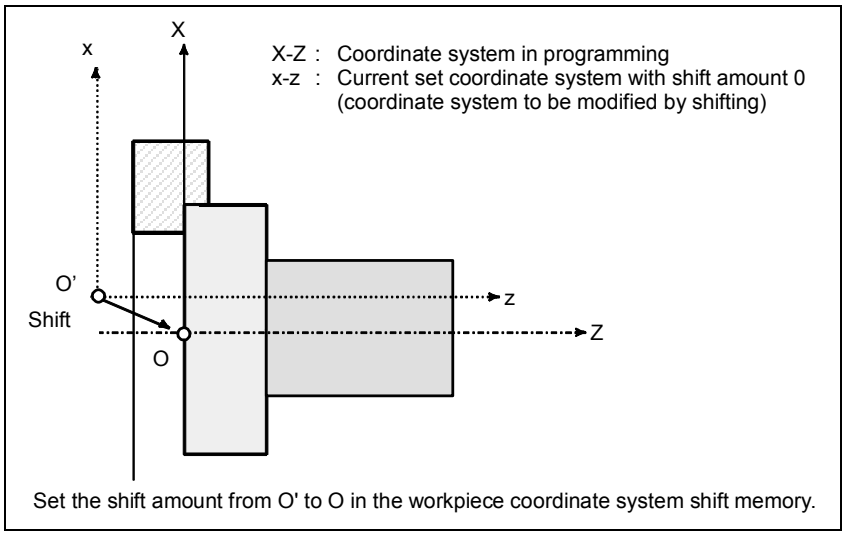


Fig. 7.2.7 (a) Workpiece coordinate system shift

Format

- Changing the workpiece coordinate system shift amount

G10 P0 IP_;

IP：Settings of an axis address and a workpiece coordinate system shift amount

> ⚠ **CAUTION**
> A single block can contain a combination of X, Y, Z, C, U, V, W, and H (in G code system A). In this case, if commands are specified for the same axis, whichever appears later becomes valid.

Limitation

- Shift amount and coordinate system setting command

Specifying a coordinate system setting command (G50 or G92) invalidates the shift amount that has already been set.

Example)

When G50X100.0Z80.0; is specified, a coordinate system is set so that the current base position of the tool is at X =100.0 and Z = 80.0, regardless of which value has been set for the workpiece coordinate system shift amount.

- Shift amount and coordinate system setting

After a shift amount is set, when automatic coordinate system setting is performed upon manual reference position return, the set coordinate system is immediately shifted by the set amount.

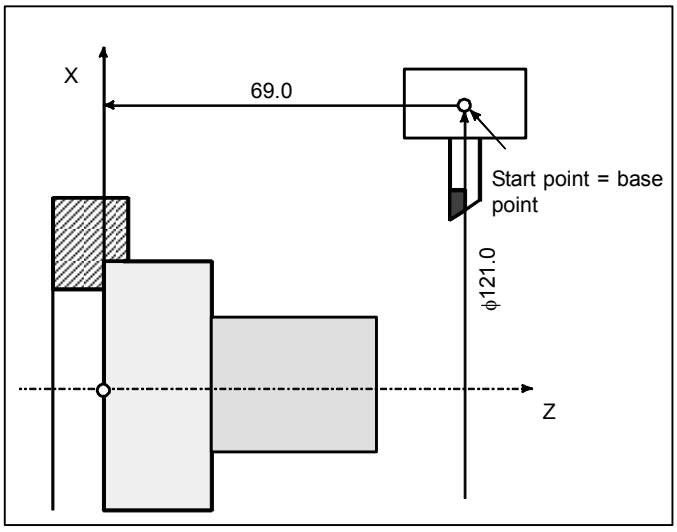
- Diameter and radius values

The workpiece coordinate system shift amount depends on diameter programming or radius programming.

Example)

Although the base point should be positioned at X = ϕ120.0 (diameter value) and Z = 70.0 from the workpiece origin, the actual position is at X = ϕ121.0 and Z = 69.0 from the origin. Set a shift amount as shown below:

X=1.0, Z=-1.0



7.3 LOCAL COORDINATE SYSTEM

When a program is created in a workpiece coordinate system, a child workpiece coordinate system can be set for easier programming. Such a child coordinate system is referred to as a local coordinate system.

Format

G52 IP_; Setting the local coordinate system
:
G52 IP 0 ; Canceling of the local coordinate system
IP_ : Origin of the local coordinate system

Explanation

By specifying G52 IP_;, a local coordinate system can be set in all the workpiece coordinate systems (G54 to G59). The origin of each local coordinate system is set at the position specified by IP_ in the workpiece coordinate system.
Once a local coordinate system is established, the coordinates in the local coordinate system are used in an axis shift command. The local coordinate system can be changed by specifying the G52 command with the origin of a new local coordinate system in the workpiece coordinate system.
To cancel the local coordinate system or specify the coordinate value in the workpiece coordinate system, match the origin of the local coordinate system with that of the workpiece coordinate system.

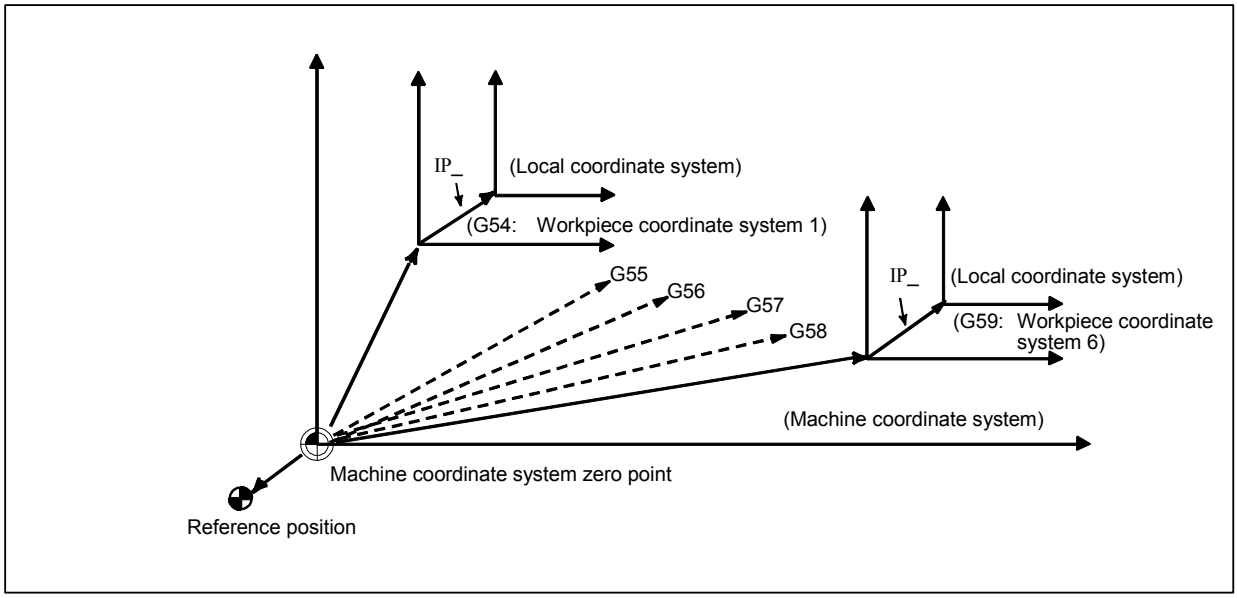


Fig. 7.3 (a) Setting the local coordinate system

⚠ CAUTION
1 When bit 2 (ZCL) of parameter No.1201 is set to 1 and an axis returns to the reference position by the manual reference position return function, the origin of the local coordinate system of the axis matches that of the workpiece coordinate system. The same is true when the following command is issued:
G52α0;
α: Axis which returns to the reference position
2 The local coordinate system setting does not change the workpiece and machine coordinate systems.

⚠ CAUTION

3 Whether the local coordinate system is canceled at reset depends on the parameter setting. The local coordinate system is canceled when either bit 6 (CLR) of parameter No.3402 or bit 3 (RLC) of parameter No.1202 is set to 1. In 3-dimensional coordinate conversion mode, however, the local coordinate system is not canceled when bit 2 (D3R) of parameter No. 5400 is set to 1.

4 When a workpiece coordinate system is set with the G92 (G50 for G code system A in the T series) command, the local coordinate system is canceled. However, the local coordinate system of an axis for which no coordinate system is specified in a G92 (G50 for G code system A in the T series) block remains unchanged.

5 G52 cancels the offset temporarily in tool radius · tool nose radius compensation.

6 Command a move command immediately after the G52 block in the absolute mode.

7.4 PLANE SELECTION

Select the planes for circular interpolation, cutter compensation, and drilling by G-code.
The following table lists G-codes and the planes selected by them.

Explanation

Table 7.4 (a) Plane selected by G code

G code	Selected plane	Xp	Yp	Zp
G17	Xp Yp plane	X-axis or an axis parallel to it	Y-axis or an axis parallel to it	Z-axis or an axis parallel to it
G18	Zp Xp plane			
G19	Yp Zp plane			

Xp, Yp, Zp are determined by the axis address appeared in the block in which G17, G18 or G19 is commanded.
When an axis address is omitted in G17, G18 or G19 block, it is assumed that the addresses of basic three axes are omitted.
Parameter No. 1022 is used to specify that an optional axis be parallel to the each axis of the X-, Y-, and Z-axes as the basic three axes.
The plane is unchanged in the block in which G17, G18 or G19 is not commanded.
The movement instruction is irrelevant to the plane selection.

M

When the power is turned on or the CNC is reset, G17 (XY plane), G18 (ZX plane), or G19 (YZ plane) is selected by bits 1 (G18) and 2 (G19) of parameter No. 3402).

T

When the power is turned on, G18 (ZX plane) is selected.
By setting bit 0 (TPS) of parameter No. 3458 to 1, it is possible to select a modal plane by setting bits 1 (G18) and 2 (G19) of parameter No. 3402.

NOTE

U-, V-, and W-axes can be used with G-codes B and C.

Example

Plane selection when the X-axis is parallel with the U-axis.

```
G17  X_Y_ ;   XY plane,
G17  U_Y_ ;   UY plane
G18  X_Z_ ;   ZX plane
     X_Y_ ;   Plane is unchanged (ZX plane)
G17 ;         XY plane
G18 ;         ZX plane
G17 U_ ;      UY plane
G18Y_ ;       ZX plane, Y axis moves regardless without any relation to the plane.
```

7.5 PLANE CONVERSION FUNCTION

Outline

This function converts a machining program created on the G17 plane in the right-hand Cartesian coordinate system to programs for other planes specified by G17.1Px commands, so that the same figure appears on each plane when viewed from the directions indicated by arrows shown in Fig.7.5 (a)

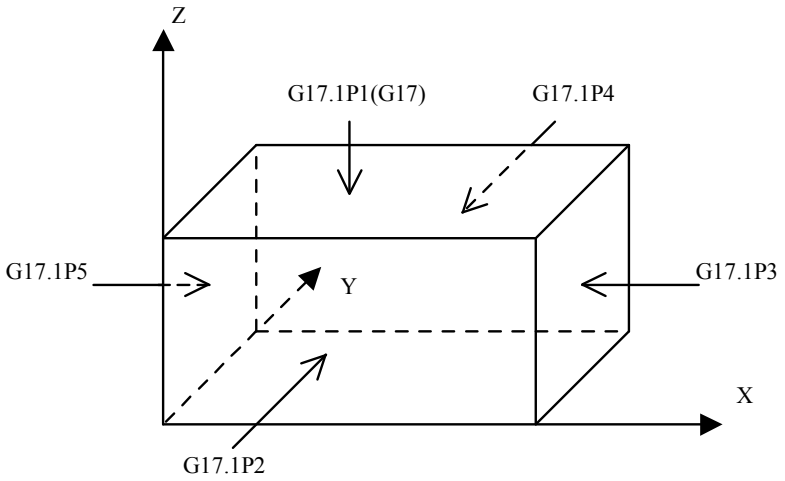


Fig.7.5 (a) Planes specified by G17.1Px commands

Format

G17.1 P_ ;

P_ : P1 to P5 Plane conversion specification

G17.1P1 is the same as G17.

Explanation

The plane conversion for a machining figure on the G17 plane shown in Fig.7.5 (b) is performed as Fig.7.5 (c), Fig.7.5 (d), Fig.7.5 (e), Fig.7.5 (f) or Fig.7.5 (g).

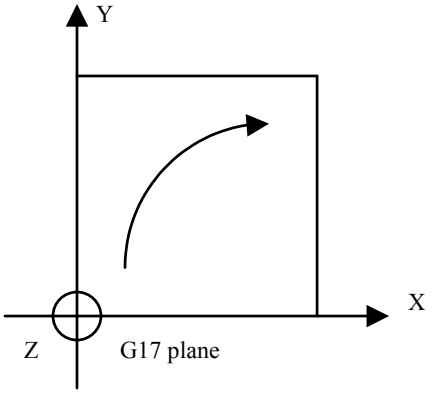


Fig.7.5 (b) G17

The circle at the origin indicates that the positive direction of the axis perpendicular to this page is the direction coming out of the page (in this case, the Z-axis is perpendicular to the XY plane).

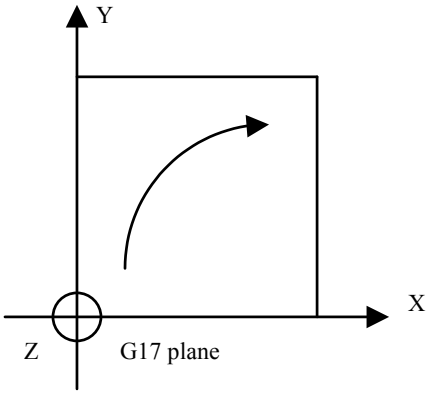


Fig.7.5 (c) G17.1P1

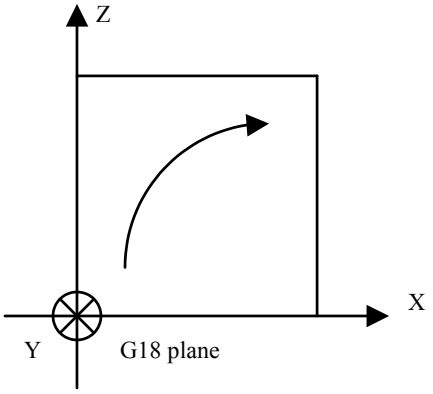


Fig.7.5 (d) G17.1P2

The cross at the origin indicates that the negative direction of the axis perpendicular to this page is the direction coming out the page (in this case, the Y-axis is perpendicular to the XZ plane).

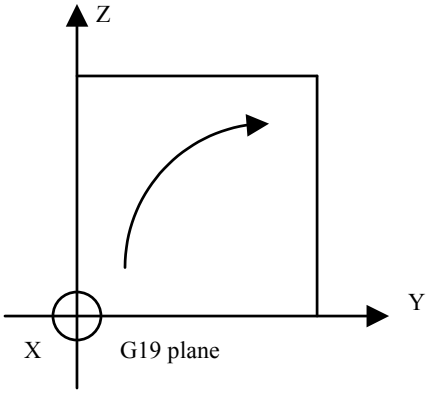


Fig.7.5 (e) G17.1P3

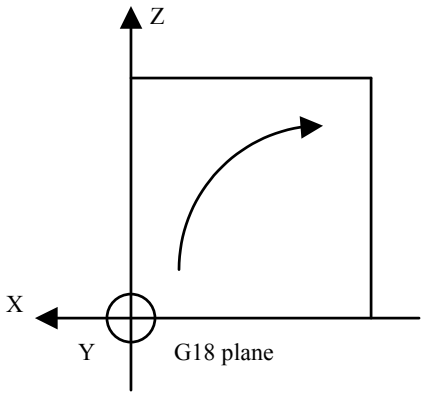


Fig.7.5 (f) G17.1P4

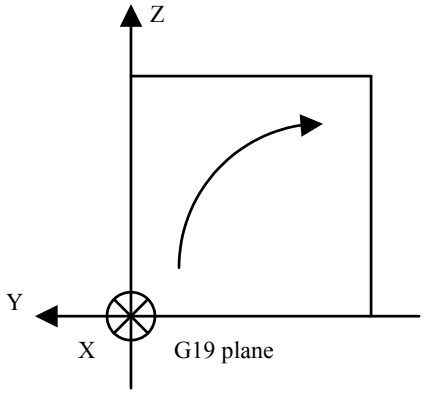


Fig.7.5 (g) G17.1P5

Program commands on the G17 plane are converted to the following commands by plane conversion:

Table7.5 (b) Program commands converted by plane conversion

Command	Plane conversion				
	G17.1P1	G17.1P2	G17.1P3	G17.1P4	G17.1P5
X	X	X	Y	-X	-Y
Y	Y	Z	Z	Z	Z
Z	Z	-Y	-X	Y	-X
G02	G02	G03	G02	G02	G03
G03	G03	G02	G03	G03	G02

Command	Plane conversion				
	G17.1P1	G17.1P2	G17.1P3	G17.1P4	G17.1P5
I	I	I	J	-I	-J
J	J	K	K	K	K
K	K	-J	I	J	-I
G41	G41	G42	G41	G41	G42
G42	G42	G41	G42	G42	G41
Tool length compensation	+	-	+	+	-
Direction of coordinate rotation	+	-	+	+	-
Direction of drilling axis	+	-	+	+	-
Plane	G17	G18	G19	G18	G19

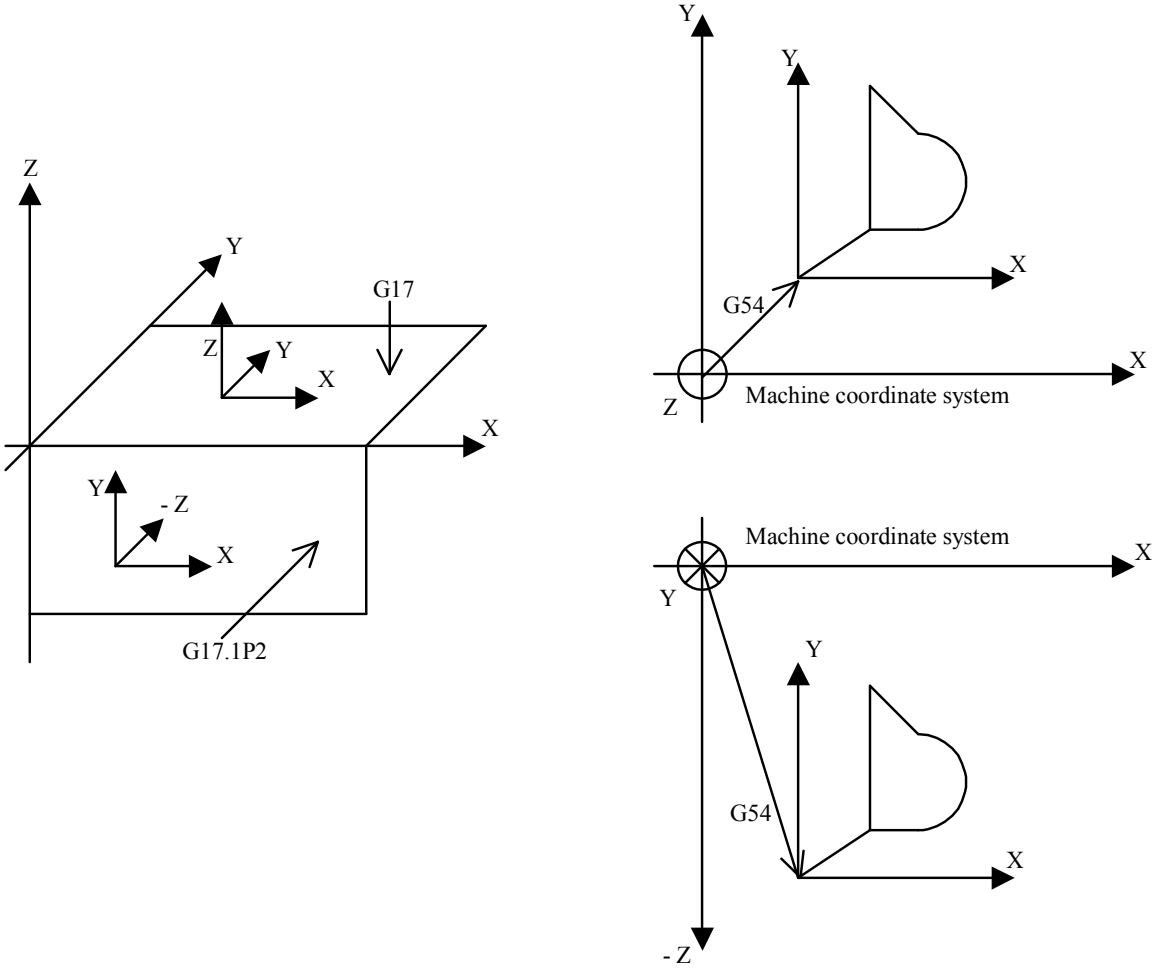
The modal information displayed in plane conversion function is as follows.

Table7.5 (c) The displayed modal information

Command	Displayed modal
G17.1P1	G17
G17.1P2	G17.1
G17.1P3	G17.1
G17.1P4	G17.1
G17.1P5	G17.1

Example

The machining program created on the G17 plane in the right-hand Cartesian coordinate system is converted to appear the same figure when viewed from the direction indicated by G17.1P2 command.




```
O1000 (MAIN PROGRAM)
N10 G91 G28 X0 Y0 Z0
N20 G54
N30 G17
N40 M98 P2000
N50 G55
N60 G17.1 P2
N70 M98 P2000
N80 G91 G28 X0 Y0 Z0
N90 M30
```

```
O2000(SUB PROGRAM)
N2010 G90 G00 Z0
N2020 G00 X0 Y0
N2030 G00 X30.0 Y20.0
N2040 G01 Z-50.0 F200
N2050 Y90.0 F500
N2060 X60.0 Y70.0
N2070 G02 Y20.0 J-25.0
N2080 G01 X30.0
N2090 G00 Z0
N2100 M99
```

Limitations

1 Plane conversion can be performed only for commands for the X-, Y-, or Z-axis.
2 Plane conversion cannot be performed for manual operation.
3 Plane conversion cannot be performed for the following commands for moving the tool to a specified
position, commands related to the machine coordinate system, and commands for setting a coordinate system:

- Automatic reference position return (G28 and G30)
- Floating reference position return (G30.1)
- Return from the reference position (G29)
- Selecting the machine coordinate system (G53)
- Stored stroke limit (G22)
- Setting the coordinate system (G54 to G59 and G92)
- Presetting the workpiece coordinate system (G92.1)
- Setting the offset (G10)

4 The current position display shows the coordinates after plane conversion (Table7.5 (d))

Table7.5 (d) Current Position Indication

Command	Absolute coordinates		
	X	Y	Z
G90 G00 X0 Y0 Z0	0.0	0.0	0.0
G17.1 P3	0.0	0.0	0.0
G00 X10.0 Y20.0	0.0	10.0	20.0
G01 Z-50.0 F200	-50.0	10.0	20.0
G02 X50.0 Y60.0 I40.0	-50.0	50.0	60.0

5 Plane conversion cannot be performed together with the axis switching function.

6 Specify plane conversion commands after canceling the following modes.
- Cutter compensation
- Tool length compensation
- Canned cycle
- Three-dimensional coordinate conversion
- Coordinate rotation
- Scaling
- Programmable mirror image

7 Plane conversion cannot be performed for the following commands that control a rotation axis together
with the X-, Y-, or Z-axis:
- Polar coordinate interpolation
- Cylindrical interpolation
- Control in normal directions
- Exponential interpolation
- Circular threading B

8 If a G17, G18, or G19 command is executed during plane conversion, the conversion is disabled and the plane specified by the command is selected.

8 COORDINATE VALUE AND DIMENSION

Chapter 8, "COORDINATE VALUE AND DIMENSION", consists of the following sections:

8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING 177
8.2 INCH/METRIC CONVERSION (G20, G21).....179
8.3 DECIMAL POINT PROGRAMMING 182
8.4 DIAMETER AND RADIUS PROGRAMMING 183
8.5 DYNAMIC SWITCHING OF DIAMETER/RADIUS SPECIFICATION 184

8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING

There are two ways to command travels of the tool; the absolute programming, and the incremental programming. In the absolute programming, coordinate value of the end position is programmed. The incremental programming is used to program the amount of a tool movement.

M

G90 and G91 are used to programming absolute or incremental programming, respectively.

T

Absolute programming or incremental programming is used depending on the programming used. See following tables.

G code system	A	B or C
Command method	Address word	G90, G91

Format

M

Absolute programming **G90 IP_ ;**
Incremental programming **G91 IP_ ;**

T

- **G code system A**

	Absolute programming	Incremental programming
X axis move command	X	U
Z axis move command	Z	W
Y axis move command	Y	V
C axis move command	C	H

- **G code system B or C**

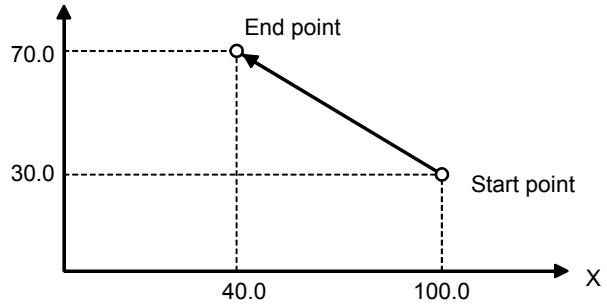
Absolute programming **G90 IP_ ;**
Incremental programming **G91 IP_ ;**

Example

M

G90 X40.0 Y70.0 ; Absolute programming

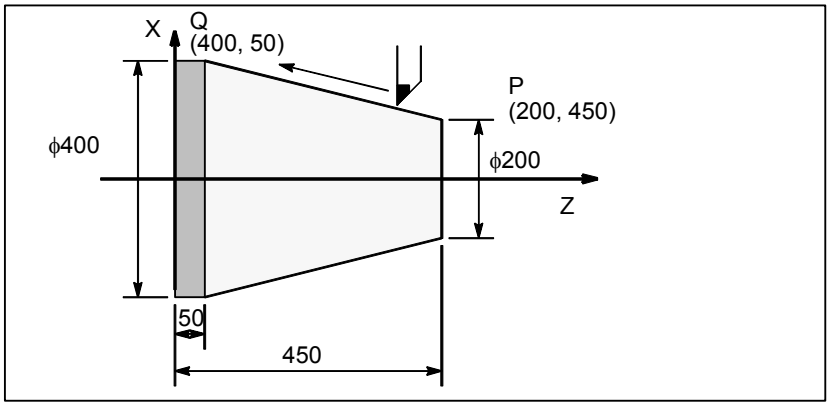
G91 X-60.0 Y40.0 ; Incremental programming



T

Tool movement from point P to point Q (diameter programming is used for the X-axis)

	G code system A	G code system B or C
Absolute programming	X400.0 Z50.0 ;	G90 X400.0 Z50.0 ;
Incremental programming	U200.0 W-400.0 ;	G91 X200.0 Z-400.0 ;



NOTE

1 Absolute programming and incremental programming can be used together in a block.
In the above example, the following command can be specified : X400.0 W-400.0 ; (in the G code system A)
2 When both X and U or Z and W are used together in a block, the one specified later is effective.
3 Incremental programming cannot be used when names of the axes are A and B during G code system A is selected.

8.2 INCH/METRIC CONVERSION (G20, G21)

Either inch or metric input (least input increment) can be selected by G code.

Format

G20 ;	Inch input
G21 ;	Metric input

This G code must be specified in an independent block before setting the coordinate system at the beginning of the program. After the G code for inch/metric conversion is specified, the unit of input data is switched to the least inch or metric input increment of increment system (see Section, "INCREMENT SYSTEM"). The unit of data input for degrees remains unchanged. The unit systems for the following values are changed after inch/metric conversion:

- Feedrate commanded by F code
- Positional command
- Workpiece origin offset value
- Tool compensation value
- Unit of scale for manual pulse generator
- Movement distance in incremental feed
- Some parameters

When the power is turned on, the G code is the same as that held before the power was turned off.

⚠ WARNING
G20 and G21 must not be switched during a program.

NOTE
1 When the least input increment and the least command increment systems are different, the maximum error is half of the least command increment. This error is not accumulated.
2 The inch and metric input can also be switched using settings (see Subsection, "Displaying and Entering Setting Data").
3 If a function selected using bit 2 (IRF) of parameter No. 14000 or bit 0 (NIM) of parameter No. 11222 is not used, be sure to perform inch/metric conversion at the reference position (machine coordinate system origin).

Performing inch/metric conversion in the reference position (parameter No. 1240 is not 0)

Conventionally, inch/metric conversion must be performed at the machine coordinate system origin. However, setting bit 2 (IRF) of parameter No. 14000 to 1 enables inch/metric conversion to be performed in the reference position (parameter No. 1240).

If an attempt is made to perform inch/metric conversion when an axis with this function enabled is not in the reference position, alarm PS5362, "CONVERT INCH/MM AT REF-POS" is issued to cancel the attempt.
Before trying to perform inch/metric conversion, be sure to set the axis of interest to the reference position, using the G28 command, for example.

In addition, if the workpiece coordinate system has been shifted from the machine coordinate system, using the following commands or operations, bit 1 (CIM) of parameter No. 11222 can be used to select whether to issue alarm PS1298 or to clear the offset.

- Manual intervention performed with the manual absolute signal being off
- Move command issued with the machine locked
- Move command issued using a handle interrupt
- Mirror image-based operation
- Workpiece coordinate system shift caused by local coordinate system setting (G52) or workpiece coordinate system setting (G92)

Switching conditions

Performing inch/metric conversion in any position other than the reference position requires satisfying all of the following conditions. Failing to satisfy any of the conditions results in alarm PS1298, "ILLEGAL INCH/METRIC CONVERSION" being issued. For electronic gear box 2 pair synchronization or electronic gear box synchronization, alarm PS1595, "ILL-COMMAND IN EGB MODE" is issued.

- Positioning or linear interpolation
- Polar coordinate interpolation cancel mode
- Polar coordinate command cancel
- Spindle Speed fluctuation detection off
- Tool radius · tool nose radius compensation cancel / 3-dimensional tool compensation cancel
- Normal direction control cancel
- Tool length offset cancel
- Scaling cancel
- Programmable mirror image cancel
- Polygon turning cancel
- Macro modal call A/B cancel
- Coordinate system rotation or 3-dimensional coordinate system conversion mode off
- Mirror image for double turret or balanced cutting mode cancel (T series only)
- Canned cycle cancel
- Electronic gear box 2 pair synchronization cancel
- Electronic gear box synchronization cancel
- Constant surface speed control cancel

In addition, performing inch/metric conversion in any position other than the reference position requires the following options.

- Inch/metric conversion
- Workpiece coordinate system

Restrictions

When performing the following operations, make sure that the axis of interest is in the reference position (machine coordinate system origin).

- Inch/metric conversion based on bit 2 (INI) of setting parameter No. 0
- Inch/metric conversion based on programmable parameter input (G10)
- Inch/metric conversion based on custom macro variable No. 3005

Performing inch/metric conversion in positions other than the reference position

Setting bit 0 (NIM) of parameter No. 11222 enables inch/metric conversion to be performed even in positions other than the reference position.

In addition, if the workpiece coordinate system has been shifted from the machine coordinate system, using the following commands or operations, bit 1 (CIM) of parameter No. 11222 can be used to select whether to issue alarm PS1298 or to clear the offset.

- Manual intervention performed with the manual absolute signal being off
- Move command issued with the machine locked

- Move command issued using a handle interrupt
- Mirror image-based operation
- Workpiece coordinate system shift caused by local coordinate system setting (G52) or workpiece coordinate system setting (G92)

If an axis is under any of the following controls, however, no automatic coordinate system conversion based on this function can be carried out for the axis.

- Axis control by PMC
- Axis synchronous control (for slave axes when the master axis is a PMC axis)
- Flexible synchronous control (for slave axes when the master axis is a PMC axis)
- Chopping
- Spindle control with servo motor
- Spindle control with Cs contour control
- Servo/spindle synchronous control

Switching conditions

Performing inch/metric conversion in any position other than the reference position requires satisfying all of the following conditions. Failing to satisfy any of the conditions results in alarm PS1298 being issued. For electronic gear box 2 pair synchronization or electronic gear box synchronization, alarm PS1595 is issued.

- Positioning or linear interpolation
- Polar coordinate interpolation cancel mode
- Polar coordinate command cancel
- Spindle Speed fluctuation detection off
- Tool radius · tool nose radius compensation cancel / 3-dimensional tool compensation cancel
- Normal direction control cancel
- Tool length offset cancel
- Scaling cancel
- Programmable mirror image cancel
- Polygon turning cancel
- Macro modal call A/B cancel
- Coordinate system rotation or 3-dimensional coordinate system conversion mode off
- Mirror image for double turret or balanced cutting mode cancel (T series only)
- Canned cycle cancel
- Electronic gear box 2 pair synchronization cancel
- Electronic gear box synchronization cancel
- Constant surface speed control cancel

In addition, performing inch/metric conversion in any position other than the reference position requires the following options.

- Inch/metric conversion
- Workpiece coordinate system
- Workpiece coordinate system preset

Restrictions

When performing the following operations, make sure that the axis of interest is in the reference position (machine coordinate system origin).

- Inch/metric conversion based on bit 2 (INI) of setting parameter No. 0
- Inch/metric conversion based on programmable parameter input (G10)
- Inch/metric conversion based on custom macro variable No. 3005

8.3 DECIMAL POINT PROGRAMMING

Numerical values can be entered with a decimal point. A decimal point can be used when entering a distance, time, or speed. Decimal points can be specified with the following addresses:

M

X, Y, Z, U, V, W, A, B, C, I, J, K, Q, R, F

T

X, Y, Z, U, V, W, A, B, C, I, J, K, R, F

Explanation

There are two types of decimal point notation: calculator-type notation and standard notation.
When calculator-type decimal notation is used, a value without decimal point is considered to be specified in millimeters inch, or deg. When standard decimal notation is used, such a value is considered to be specified in least input increments. Select either calculator-type or standard decimal notation by using the bit 0 (DPI) of parameter No.3401.Values can be specified both with and without decimal point in a single program.

Example

Program command	Pocket calculator type decimal point programming	Standard type decimal point programming
X1000 Command value without decimal point	1000mm Unit :mm	1mm Unit : Least input increment (0.001mm)
X1000.0 Command value with decimal point	1000mm Unit :mm	1000mm Unit :mm

⚠ CAUTION
When specifying a dimension word for a command G code in a block, be sure to place the dimension word after the command G code.

NOTE
1 A specified value less than the minimum unit is treated as described below.
Example 1)
When a value is specified directly at an address (in the case of 1/1000mm IS-B)
X1.2345 ; Treated as X1.235
X-1.2345 ; Treated as X-1.234
Example 2)
When a value is assigned to a macro variable (in the case of 1/1000mm IS-B)
#100=1.2345;
X#100 ; Treated as X1.235
#100=-1.2345;
X#100 ; Treated as X-1.234

NOTE

2 When more than eight digits are specified, an alarm occurs. If a value is entered with a decimal point, the number of digits is also checked after the value is converted to an integer according to the least input increment.

Examples:

X1.23456789;

Alarm PS0003, “TOO MANY DIGIT” occurs because more than eight digits are specified.

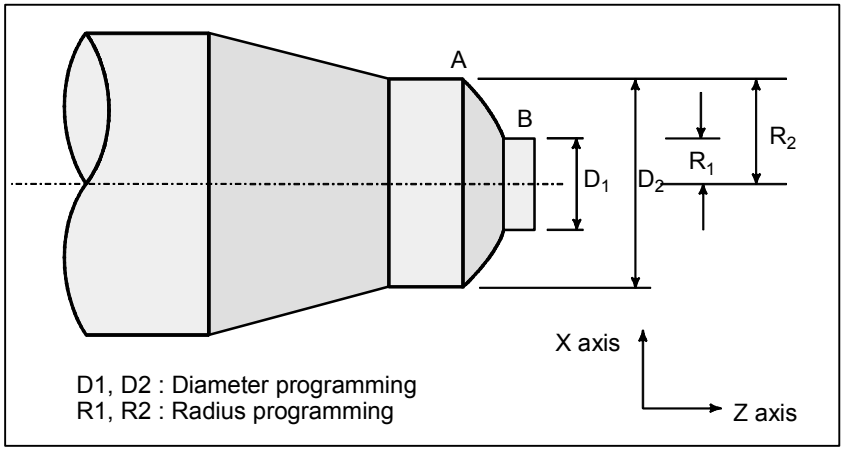
X123456.7;

If the least input increment is 0.001 mm, the value is converted to integer 123456700. Because the integer has more than eight digits, an alarm occurs.

8.4 DIAMETER AND RADIUS PROGRAMMING

Since the workpiece cross section is usually circular in CNC lathe control programming, its dimensions can be specified in two ways :

Diameter and Radius



When the diameter is specified, it is called diameter programming and when the radius is specified, it is called radius programming.

Explanation

- Notes on diameter programming/radius programming for each command

Radius programming or diameter programming can be specified by bit 3 (DIA) of parameter No.1006. When using diameter programming, note the conditions listed in the Table 8.4 (a).

Table 8.4 (a) Notes on specifying diameter value

Item	Notes
X axis command	Specified with a diameter value
Incremental command	Specified with a diameter value In the above figure, specifies D2 minus D1 for tool path B to A.
Coordinate system setting (G50)	Specifies a coordinate value with a diameter value
Component of tool offset value	Bit 1 of parameter No.5004 determines either diameter or radius value
Parameters in canned cycle, such as cutting depth along X axis. (R)	Specifies a radius value
Radius designation in circular interpolation (R, I, K, and etc.)	Specifies a radius value
Feedrate along axis	Specifies change of radius/rev. or change of radius/min.

Item	Notes
Display of axis position	Displayed as diameter value

8.5 DIAMETER AND RADIUS SETTING SWITCHING FUNCTION

Overview

Usually, whether to use diameter specification or radius specification to specify a travel distance on each axis is uniquely determined by the setting of bit 3 (DIAx) of parameter No. 1006. However, this function enables switching between diameter specification and radius specification by using a signal or G code.
Thus, a coordinate, program, and so forth can be specified by switching between diameter specification and radius specification for each controlled axis.

Explanation

- Selection of a diameter/radius specification switching method

Two methods are available for switching between diameter specification and radius specification:
1) Signal
2) G code
Use bit 5 (PGD) of parameter No. 3400 to determine which method to use.

- Switching method using a signal

For switching between diameter specification and radius specification, set the diameter/radius specification switch signal from DI1 to DI8 (input signals) corresponding to a desired axis, from 0 to 1.
If an input signal is set from 0 to 1, and radius specification is selected (with bit 3 (DIAx) of parameter No. 1006 = 0) for the axis corresponding to the input signal, the specification method switches to diameter specification; the specification method switches to radius specification if diameter specification is selected (with bit 3 (DIAx) of parameter No. 1006 = 1).
During switching, the diameter/radius specification switching in-progress signal from DM1 to DM8 (output signals) corresponding to a switched axis is output.
To return the diameter/radius specification of an axis to the original state, set the setting of the corresponding diameter/radius specification switch signal from DI1 to DI8, from 1 to 0.

NOTE
1 When operating an input signal by using an M code, for example, during automatic operation, perform a switching operation according to the method below to reflect the state of diameter/radius specification switching in the execution block correctly.
As an auxiliary function for switching, use an unbuffered M code (parameter No. 3411 and up). Use the following sequences for a specified M code:
- When switching is performed
  M code → Input signal ON → Confirmation of output signal ON → FIN
- When switching is cancelled
  M code → Input signal OFF → Confirmation of output signal OFF → FIN

If a diameter/radius specification switch signal is operated during automatic operation without following the sequences above, alarm PS5320, “DIA./RAD. MODE CAN’T BE SWITCHED.” is issued.
2 If a diameter/radius specification switch signal is operated while a movement is made on an axis subject to switching, alarm PS5320 is issued.

- Switching method using a G code (programmable diameter/radius specification switching)

The format of a G code for diameter/radius specification switching is as follows:

Format

G10.9 IP_ ;

IP_ : Address and command value of an specified axis subject to diameter/radius specification switching
Specify 0 or 1 as the command value.
0: Radius specification
1: Diameter specification

NOTE
1 Specify G10.9 in a single block specifying no other codes.
2 After an axis address, specify a command value without using the decimal point.

- Switching operation

According to the switching methods above, diameter/radius specification is internally switched as described below.

1) Switching using a signal
   - When parameter DIAx = 0 (radius specification) → Operation is performed with radius specification.
   - When parameter DIAx = 1 (diameter specification) → Operation is performed with diameter specification.
2) Switching using a G code
   - When the specified address value = 0 (radius specification) → Operation is performed with radius specification.
   - When the specified address value = 1 (diameter specification) → Operation is performed with diameter specification.

NOTE
1 When the diameter/radius specification switching state is to be cancelled using a reset or mode switching at the time of signal-based switching, the input signal needs to be operated.
2 Switching using a G code is cancelled by a reset.

⚠ CAUTION
When switching is performed from diameter specification to radius specification, the travel distance based on the same move command is doubled when compared with diameter specification. So, when switching from diameter specification to radius specification, ensure safety in machine operation.

- Selecting the machine coordinate system

When diameter and radius setting switching is taking place as directed with the diameter and radius setting switching function, the coordinates specified by a machine coordinate system select command (G53) vary depending on the value of bit 7 (PDM) of parameter No. 11222. If the parameter PDM is 0, diameter and radius switching occurs. If the parameter PDM is 1, the coordinates vary according to the value of bit 3 (DIAx) of parameter No. 1006.

Limitation

- Feedrate

A radius-based feedrate is specified in both of diameter specification and radius specification at all times.

- Data not switchable

The following data follows the setting of parameter DIAx, so that diameter and radius switching is not performed:

- Parameter
- Offset
- Workpiece coordinate system
- Scale display on the graphic screen

NOTE
For offset data, the settings of bit 1 (ORC) of parameter No. 5004 and bit 2 (ODI) of parameter No. 5004 have priority.

- Switchable data and commands

For the following data and commands, diameter and radius switching is performed according to the specified specification method:

- Programmed move command
- Current position display
- Workpiece coordinate system preset
- Movement based on the manual numeric command G00 or G01

- Use with other functions

Diameter/radius specification switching cannot be performed for an axis on which a movement is being made with any of the functions indicated below.
Moreover, none of the functions indicated below can be performed during diameter and radius switching.

- Synchronous/composite control
- Superimposed Control
- Axis synchronous control
- PMC axis control

9 SPINDLE SPEED FUNCTION (S FUNCTION)

The spindle speed can be controlled by specifying a value following address S.
Chapter 9, "SPINDLE SPEED FUNCTION (S FUNCTION)", consists of the following sections:

9.1 SPECIFYING THE SPINDLE SPEED WITH A CODE 187
9.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5-DIGIT COMMAND) 187
9.3 CONSTANT SURFACE SPEED CONTROL (G96, G97) 187
9.4 SPINDLE POSITIONING FUNCTION 191
9.5 SPINDLE SPEED FLUCTUATION DETECTION 196
9.6 SPINDLE CONTROL WITH SERVO MOTOR 199

9.1 SPECIFYING THE SPINDLE SPEED WITH A CODE

When a value is specified after address S, the code signal and strobe signal are sent to the machine to control the spindle rotation speed.
A block can contain only one S code. Refer to the appropriate manual provided by the machine tool builder for details such as the number of digits in an S code or the execution order when a move command and an S code command are in the same block.

9.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5-DIGIT COMMAND)

The spindle speed can be specified directly by address S followed by a max. five-digit value (min^{-1}). The unit for specifying the spindle speed may vary depending on the machine tool builder. Refer to the appropriate manual provided by the machine tool builder for details.

9.3 CONSTANT SURFACE SPEED CONTROL (G96, G97)

Specify the surface speed (relative speed between the tool and workpiece) following S. The spindle is rotated so that the surface speed is constant regardless of the position of the tool.

Format

- Constant surface speed control command

G96S<u>xxxxx</u> ;

↑ Surface speed (m/min or feet/min)
This surface speed unit may change according to machine tool builder's specification.

- Constant surface speed control cancel command

G97S<u>xxxxx</u> ;

↑ Spindle speed (min^{-1})
This surface speed unit may change according to machine tool builder's specification.

- Constant surface speed controlled axis command

G96Pα ;

P0 : Axis set in the parameter No. 3770
P1 : 1st axis, P2 : 2nd axis, P3 : 3rd axis, P4 : 4th axis
P5 : 5th axis, P6 : 6th axis, P7 : 7th axis, P8 : 8th axis

NOTE
If multi-spindle control (spindle selecting based on address P or spindle name expansion) is enabled, axis specification based on address P is disabled. Use parameter No. 3770 for axis specification.

- Clamp of maximum spindle speed

G92 S_ ;

The maximum spindle speed (min^{-1}) follows S.

T

G50 S_ ;

The maximum spindle speed (min^{-1}) follows S.

NOTE
G50 can be used with G code system A.

Explanation

- Constant surface speed control command (G96)

G96 (constant surface speed control command) is a modal G code. After a G96 command is specified, the program enters the constant surface speed control mode (G96 mode) and specified S values are assumed as a surface speed. A G96 command must specify the axis along which constant surface speed control is applied. A G97 command cancels the G96 mode. When constant surface speed control is applied, a spindle speed higher than the value specified in G92S_; or G50S_; (maximum spindle speed) is clamped at the maximum spindle speed. When the power is turned on, the maximum spindle speed is not yet set and the speed is not clamped. S (surface speed) commands in the G96 mode are assumed as S = 0 (the surface speed is 0) until M03 (rotating the spindle in the positive direction) or M04 (rotating the spindle in the negative direction) appears in the program.

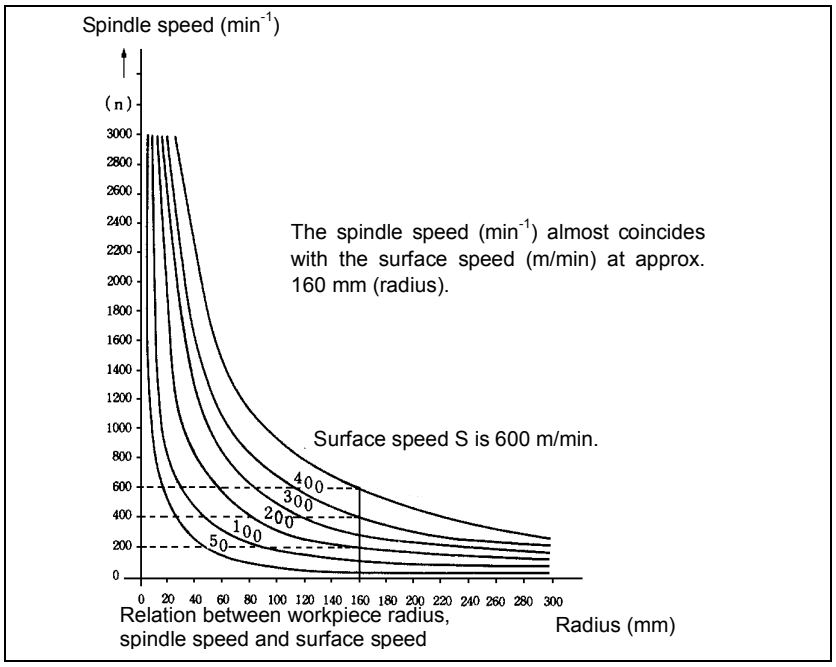


Fig. 9.3 (a) Relation between workpiece radius, spindle speed and surface speed

- Setting the workpiece coordinate system for constant surface speed control

To execute the constant surface speed control, it is necessary to set the workpiece coordinate system , and so the coordinate value at the center of the rotary axis, for example, Z axis, (axis to which the constant surface speed control applies) becomes zero.

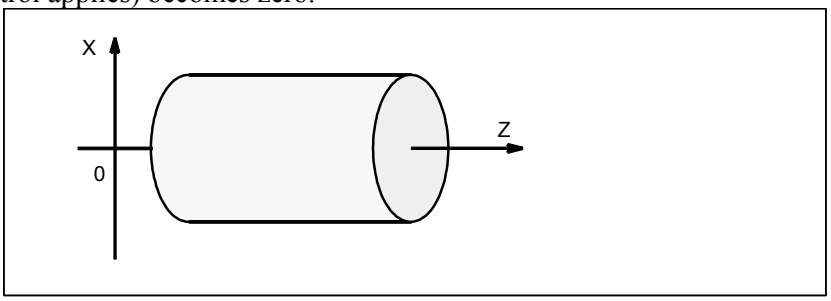
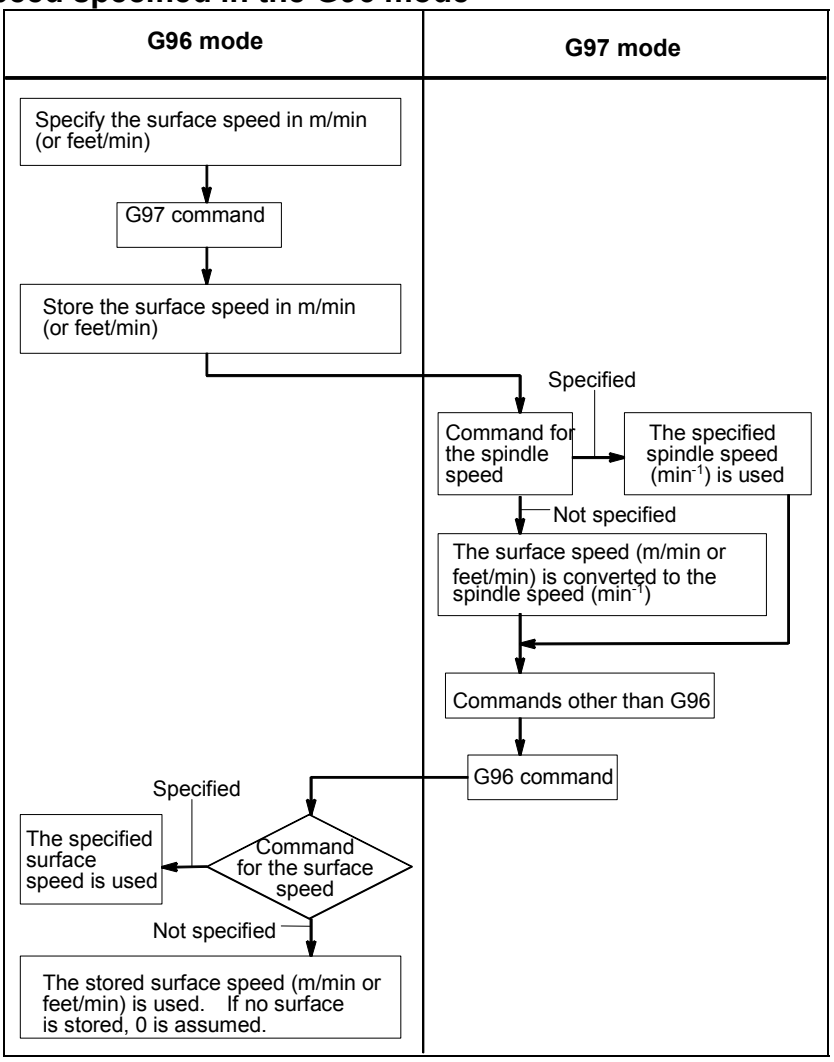


Fig. 9.3 (b) Example of the workpiece coordinate system for constant surface speed control

- Surface speed specified in the G96 mode



- Maximum spindle speed clamp

If the spindle rotating at the maximum speed specified with the relevant parameter may cause a problem in the constant surface speed control mode, execute maximum spindle speed clamp (M series: G92S_;, T series: G50S_;) before starting constant surface speed control so that the spindle speed does not exceed the specified speed.

When maximum spindle speed clamp has not been executed after power-on, constant surface speed control may be specified due to a program error or operator error including the use of an improper procedure for executing a machining program. To avoid this problem, bit 4 (CSA) of parameter No. 3712 can be set to 1 to issue alarm PS5557, “NO MAX SP SPEED CLAMP COMMAND” in the above case to prevent the spindle from rotating at an unexpected speed.

Limitation

- Constant surface speed control for threading

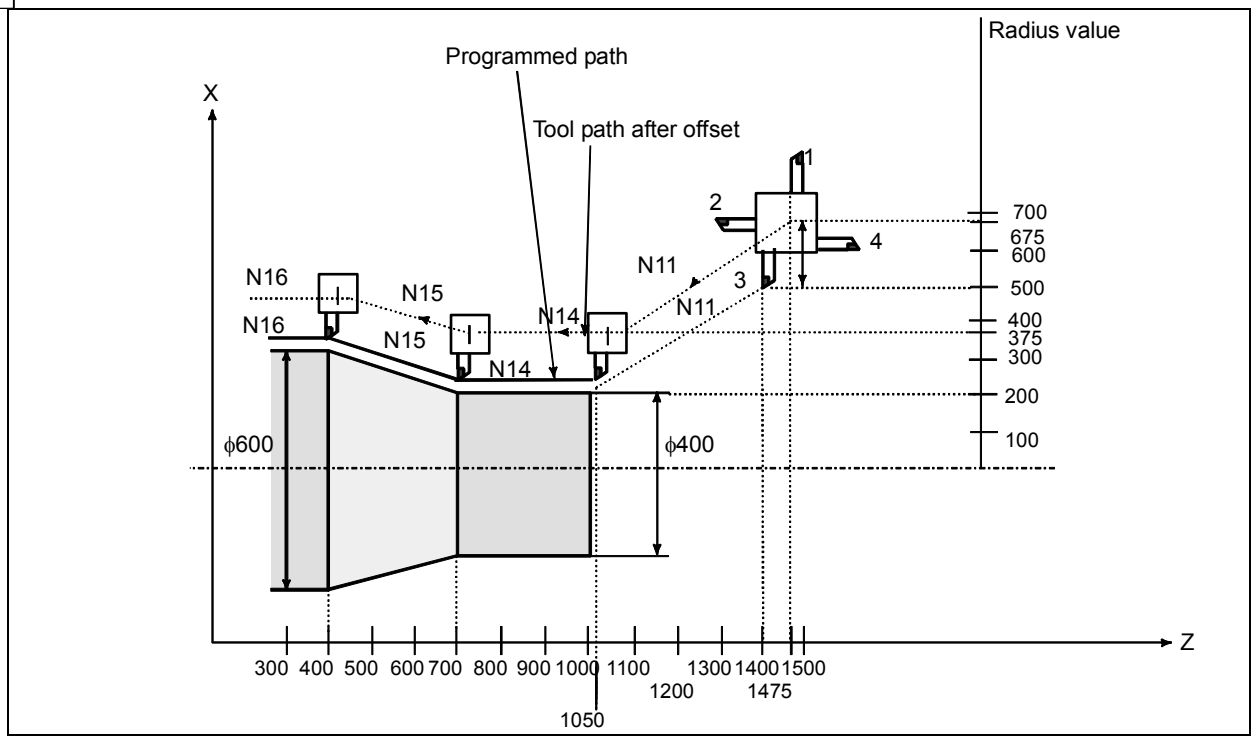
The constant surface speed control is also effective during threading. Accordingly, it is recommended that the constant surface speed control be invalidated with G97 command before starting the scroll threading and taper threading, because the response problem in the servo system may not be considered when the spindle speed changes.

- Constant surface speed control for rapid traverse (G00)

In a rapid traverse block specified by G00, the constant surface speed control is not made by calculating the surface speed to a transient change of the tool position, but is made by calculating the surface speed based on the position at the end point of the rapid traverse block, on the condition that cutting is not executed at rapid traverse.

Example

T



N8 G00 X1000.0Z1400.0 ;
N9 T33;
N11 X400.0Z1050.0;
N12 G50S3000 ; (Designation of max. spindle speed)
N13 G96S200 ; (Surface speed 200 m/min)
N14 G01 Z 700.0F1000 ;
N15 X600.0Z 400.0;
N16 Z_ ;

The CNC calculates the spindle speed which is proportional to the specified surface speed at the position of the programmed coordinate value on the X axis. This is not the value calculated according to the X axis coordinate after offset when offset is valid. At the end point N15 in the example above, the speed at 600 dia. (Which is not the turret center but the tool nose) is 200 m/min. If X axis coordinate value is negative, the CNC uses the absolute value.

9.4 SPINDLE POSITIONING FUNCTION

Overview

In turning, the spindle connected to the spindle motor is rotated at a certain speed to rotate the workpiece mounted on the spindle. This spindle control status is referred to as spindle rotation mode.

The spindle positioning function turns the spindle connected to the spindle motor by a certain angle to position the workpiece mounted on the spindle at a certain angle. This spindle control status is referred to as spindle positioning mode.

The spindle positioning function involves the following three operations:

1. Canceling the spindle rotation mode and entering the spindle positioning mode
   Place the spindle in the spindle positioning mode and establish a reference position by specifying a given M code (set with a parameter). (Spindle orientation)
2. Positioning the spindle in the spindle positioning mode
   The spindle is positioned in either of the two methods:
   1) Positioning with an arbitrary angle by an axis address
   2) Positioning with a semi-fixed angle by a given M code (set with a parameter)
3. Canceling the spindle positioning mode, and entering the spindle rotation mode
   Place the spindle in the spindle rotation mode by specifying a given M code (set with a parameter).

The least command increment, least input increment, and maximum value for the spindle positioning axis are as follows:

- Least command increment
  $\frac{360}{4096} \fallingdotseq 0.088$ deg (when the gear ratio of the spindle to the position coder
- Least input increment
  0.001 deg (IS-B)
- Maximum value
  ±999999.999 deg

9.4.1 Spindle Orientation

When spindle positioning is first performed after the spindle motor is used for normal spindle operation, or when spindle positioning is interrupted, the spindle orientation is required.
Orientation permits the spindle to stop at a predetermined position.
Orientation is directed by the M code set in parameter No. 4960. The direction of orientation can be set with a parameter.
For the serial spindle, it is set in bit 4 (RETSV) of parameter No. 4000. With the grid shift function, the orientation position can be shifted in a range of 0 to 360 deg with parameter No. 4073 for a serial spindle.

- Feedrate during spindle orientation

An orientation feedrate for a serial spindle is determined by a spindle parameter setting.
In orientation, the serial spindle stops at the orientation position after several rotations of the spindle motor.

- Omission of orientation

By using bit 2 (ISZ) of parameter No. 4950, orientation upon switching to spindle positioning mode can be omitted if it is unnecessary (for example, when no start position is specified and incremental positioning from the current position is only required). More specifically, when an M code for switching to spindle positioning mode is specified, the spindle control mode is simply switched to spindle positioning mode and then the processing is completed without orientation.

- Program reference position

The position at which orientation is completed is assumed to be a program reference position. However, the program reference position can be changed through coordinate system setting (G92 or G50) or automatic coordinate system setting (bit 0 (ZPR) of parameter No. 1201).
When a setting is made to omit orientation, a program reference position is not established, and operation by an absolute command is unpredictable during spindle positioning with an axis address.

9.4.2 Spindle Positioning

The spindle can be positioned with a semi-fixed angle or arbitrary angle.

- Positioning with a semi-fixed angle

Use an M code to specify a positioning angle. The specifiable M code value may be one of the six values from Mα to M (α+5). Value α must be set in parameter No. 4962 beforehand. The positioning angles corresponding to Mα to M (α+5) are listed below. Value β must be set in parameter No. 4963 beforehand.

M-code (Ex.) β = α + 5	Positioning angle	(Ex.) β = 30°
Mα	β	30°
M (α + 1)	2β	60°
M (α + 2)	3β	90°
M (α + 3)	4β	120°
M (α + 4)	5β	150°
M (α + 5)	6β	180°

When the number of M codes to be used, value γ, is specified in parameter No. 4964, a specifiable M code value may be in a range of values from Mα to M (α + (γ - 1)), up to 255 values from Mα to M (α + (255 - 1)).

M code (Ex.) γ = 11	Positioning angle	(Ex.) β = 30°
Mα	β	30°
M (α + 1)	2β	60°
M (α + 2)	3β	90°
M (α + 3)	4β	120°
...	...	...
M (α + 11 - 1)	11β	330°

The direction of rotation can be specified in bit 1 (IDM) of parameter No.4950.

- Positioning with an arbitrary angle

Specify the position with an arbitrary angle using the axis address followed by a signed numeric value or numeric values. The axis address must be specified in the G00 mode.
(The explanation below is given assuming that a C axis address is set.)

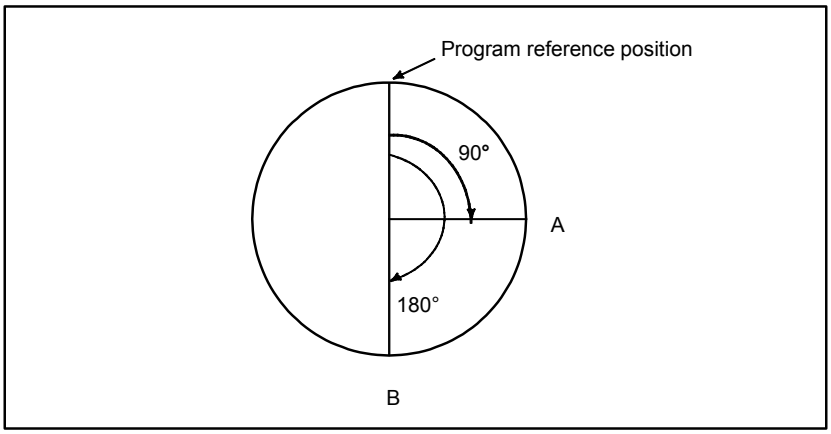
(Example) C-45000
C180.000

A numeric with the decimal point can be entered. The value must be specified in degrees.

(Example) C36.0=C36 degrees

- Absolute commands and incremental commands

Incremental commands are always used for positioning with a semi-fixed angle (using M codes).
The direction of rotation can be specified with bit 1 (IDM) of parameter No. 4950.
Absolute and incremental commands can be used for positioning with an arbitrary angle.
With absolute commands for positioning with an arbitrary angle, when the rotation axis rollover function is used (bit 0 (ROA) of parameter No. 1008 is 1), shortcut control is also enabled (bit 1 (RAB) of parameter No. 1008 is 0).



Command format		G-code system A in lathe system		G-code system B or C in lathe system, and machining center system	
		Address used	Command A-B in the above figure	Address used and G code	Command A-B in the above figure
Absolute command	Specify the end point with a distance from the program reference position.	C	C180.0 ;	G90, C	G90 C180 .;
Incremental command	Specify a distance from the start point to the end point.	H	H90.0 ;	G91, C	G91 C90 .;

- Feedrate during positioning

The feedrate during positioning equals the rapid traverse speed specified in parameter No. 1420.
For the specified speed, an override of 100%, 50%, 25%, and F0 (parameter No. 1421) can be applied.

9.4.3 Canceling Spindle Positioning

When modes are to be switched from spindle positioning to normal spindle rotation, the M code set in parameter No. 4961 must be specified.

Also, the spindle positioning mode is canceled and the spindle rotation mode is set in the following cases:

<1> A reset operation (including an emergency stop) occurs when a servo alarm is issued.
<2> A reset operation (including an emergency stop) occurs when a spindle alarm is issued.
<3> An orientation operation in progress is stopped due to a reset or alarm, or for some other reason.
<4> A reset operation (including an emergency stop) occurs when bit 0 (IOR) of parameter No. 4950 is 1.

⚠ CAUTION

1 During execution of spindle positioning sequences (canceling the spindle rotation mode and entering the spindle positioning mode, positioning the spindle in the spindle positioning mode, and canceling the spindle positioning mode and entering the spindle rotation mode), the automatic operation stop signal *SP is invalid. This means that automatic operation does not stop until all the sequences are completed, even when the *SP signal becomes “0”.
2 Dry run and machine lock cannot be performed during spindle positioning.
3 Auxiliary function lock is disabled for M codes for the spindle positioning function.
4 The serial spindle Cs-axis contour control function and the spindle positioning function cannot be used at a time. If both options are specified, the spindle positioning function has priority.

⚠ CAUTION

5 The spindle positioning axis is handled as a controlled axis. Therefore, controlled axis-related signals (such as the overtravel signal) must be set.
6 When using the rigid tapping function and the spindle positioning function concurrently, do not specify rigid tapping in spindle positioning mode or spindle positioning in rigid tapping mode.

NOTE

1 M code commands for positioning of a spindle must be specified in a single block. Other commands must not be contained in the same block. (Also, M code commands for positioning of another spindle must not be contained in the same block.)
Even when the single-block, multiple-M code command function is also used, related M codes must be specified in a single block.
2 Even when the single-block, multiple-M code command function is also used, related M codes must be specified in a single block.
3 Axis address commands for positioning of a spindle must be specified in a single block. Other commands must not be contained in the same block. However, the following commands can be contained in the same block where axis address commands are specified:
G00, G90, G91, G92 (G-code systems B and C in the T series, M series)
G00, G50 (G-code system A in the T series)
4 M code commands for spindle positioning specify M codes that are not buffered.
5 Spindle positioning cannot be performed by manual operation (in jog feed, manual handle feed, manual numeric command, or other mode).
6 Spindle positioning cannot be performed by PMC axis control.
7 For spindle positioning, program restart and block restart operations cannot be performed. Use the MDI for these operations.
8 The stored stroke limit check is disabled for the spindle positioning axis.
9 The axis removal function is disabled for the spindle positioning axis.
10 The pitch error compensation function is disabled for the spindle positioning axis.
11 When a setting is made to omit spindle orientation, the reference position return completion signal does not become “1”.
12 In spindle orientation, all-axis interlock and axis-specific interlock are checked only when a block is started. A signal is ignored if input during the execution of the block.
13 A difference between a specified travel distance and an actual travel distance is maintained until spindle positioning mode is canceled.

9.5 SPINDLE SPEED FLUCTUATION DETECTION

Overview

With this function, an overheat alarm OH0704, "OVERHEAT" is raised and the spindle speed fluctuation detection alarm signal SPAL is issued when the spindle speed deviates from the specified speed due to machine conditions.
This function is useful, for example, for preventing the seizure of the guide bushing.
G26 enables spindle speed fluctuation detection. G25 disables spindle speed fluctuation detection.

Format

- Spindle fluctuation detection on

G26 Pp Qq Rr Ii;

p: Time (in ms) from the issue of a new spindle rotation command (S command) to the start of checking whether the actual spindle speed is so fast that an overheat can occur.
When a specified speed is reached within the time period of P, a check is started at that time.

q: Tolerance (%) of a specified spindle speed

$$q = \left| 1 - \frac{actual\ spindle\ speed}{specified\ spndle\ speed} \right| \times 100$$

If a specified spindle speed lies within this range, it is regarded as having reached the specified value. Then, the checking of an actual spindle speed is started.

r: Spindle speed fluctuation (%) at which the actual spindle speed is so fast that an overheat can occur

$$r = \left| 1 - \frac{speed\ that\ can\ cause\ overheat}{speciofied\ spndle\ speed} \right| \times 100$$

If the fluctuation of the actual spindle speed to the specified spindle speed exceeds the spindle speed fluctuation of R, the actual spindle speed is regarded as being so fast that an overheat can occur.

i: Spindle speed fluctuation width at which the actual spindle speed (min^{-1}) is so fast that an overheat can occur
If the fluctuation (width) between the specified and actual spindle speeds exceeds the spindle speed fluctuation width of I, the actual spindle speed is regarded as being so fast that an overheat can occur.

G26 enables the spindle speed fluctuation detection function. The values specified for P, Q, R, and I are set in the following parameters: No. 4914, No. 4911, No. 4912, and No. 4913, respectively. Each command address corresponds to a parameter number as listed below.

Command address	Parameter number
Q	No.4911
R	No.4912
I	No.4913
P	No.4914

If the P, Q, R, or I command address is omitted, the function detects the fluctuation of the actual spindle speed according to the value set in the corresponding parameters Nos. 4914, 4911, 4912, or 4913.
The parameters Nos. 4914, 4911, 4912, and 4913 for the spindle on which the currently selected position coder is mounted are used for the setting and spindle speed fluctuation detection check.

- Spindle fluctuation detection off

G25;

G25 disables the spindle speed fluctuation detection function.
When G25 is specified, the parameters Nos. 4914, 4911, 4912, and 4913 are unchanged. When the power is turned on or after a reset (clear state (bit 6 (CLR) of parameter No. 3402 = 1)) is executed, the spindle speed fluctuation detection function is disabled (G25). For the clear state, also check the setting of bit 3 (C19) of parameter No. 3408 for the M series or bit 0 (C08) of parameter No. 3407 for the T series.

Explanation

The function for detecting spindle speed fluctuation checks whether the actual speed varies for the specified speed or not. Si or Sr, whichever is greater, is taken as the allowable fluctuation speed (Sm). An alarm (OH0704) is activated when the actual spindle speed varies for the commanded speed (Sc) under the condition that the variation width exceeds the allowable variation width (Sm).

$|Sc - Sa| > Sm$

Sc : Specified spindle speed
Sa : Actual spindle speed
Si : The allowable constant variation width which is independent of the specified spindle speed (parameter No.4913)
Sr : The allowable variation width which is obtained by multiplying Sc (commanded spindle speed) by r (constant ratio). (r = parameter No.4912)

Bit 0 (FLR) of parameter No. 4900= 0	Bit 0 (FLR) of parameter No. 4900= 1
$Sr = Sc \times \frac{r}{100}$	$Sr = Sc \times \frac{r}{1000}$

Sm: Si or Sr, whichever is greater

- Conditions to start spindle speed fluctuation detection

If the specified spindle speed Sc changes, spindle speed fluctuation detection starts when one of the conditions below is met.
When bit 7 (FDTs) of parameter No. 7900 is set to 1, spindle speed fluctuation detection starts when condition <2> is met.

<1> The actual spindle speed falls in a range of (Sc - Sq) to (Sc + Sq)
Sc : Specified spindle speed
Sq : Tolerance within which the spindle is assumed to attain the programmed speed (parameter No.4911)

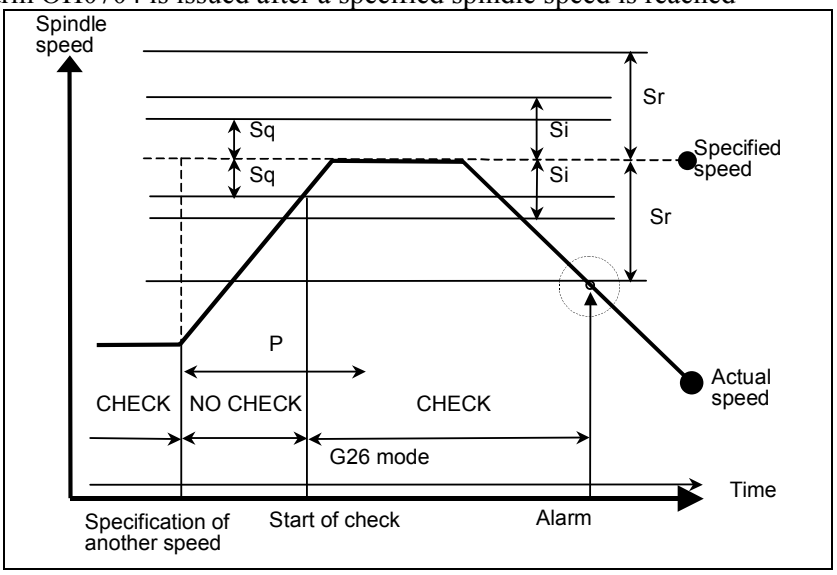
Parameter FLR= 0	Parameter FLR= 1
$Sq = Sc \times \frac{q}{100}$	$Sq = Sc \times \frac{q}{1000}$

<2> When time p specified in parameter No. 4914 elapses after the specified speed Sc changes.

- Examples of spindle speed fluctuation detection

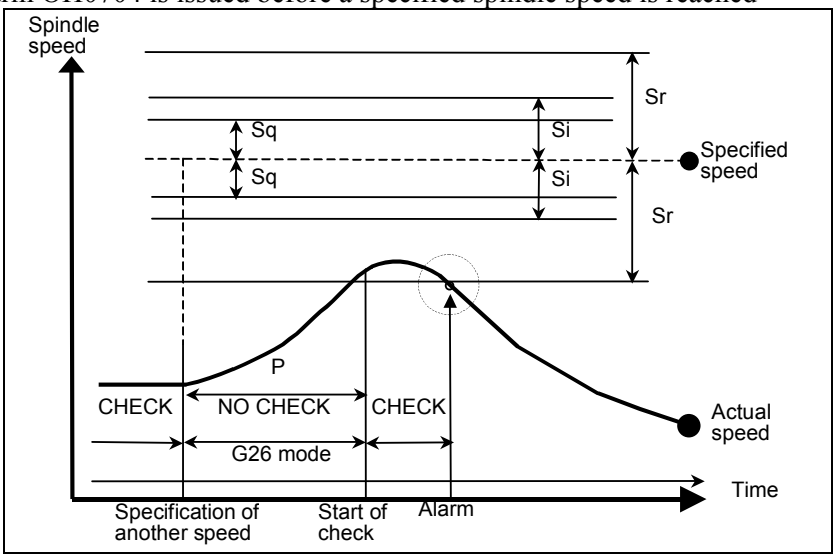
(Example 1)

When an alarm OH0704 is issued after a specified spindle speed is reached



(Example 2)

When an alarm OH0704 is issued before a specified spindle speed is reached



Specified speed : (Speed specified by address S and five-digit value) × (spindle override)
Actual speed : Speed detected with a position coder

p : Period after a change occurs in the actual spindle speed until detection starts
Parameter No.4914, address P

Sq: (Specified spindle speed) × (Detection start tolerance (q))
Parameter No.4911, address Q

Parameter FLR = 0	Parameter FLR = 1
$\frac{q}{100}$	$\frac{q}{1000}$

Sr : (Specified spindle speed) × (Allowable variation (r))
Parameter No.4912, address R

Parameter FLR = 0	Parameter FLR = 1
$\frac{r}{100}$	$\frac{r}{1000}$

Si : Allowable variation width
Parameter No.4913, address I

If the difference between the specified speed and actual speed exceeds both Sr and Si, an alarm OH0704 is raised.

- Relationship between spindle speed control and each spindle

Function \ Spindle	Serial spindle			
	1st spindle	2nd spindle	3rd spindle	4th spindle
Spindle speed fluctuation detection	Possible	Possible(*1)	Possible(*1)	Possible(*1)

NOTE
1 An optional function of multi spindle control is necessary.
2 The spindle speed fluctuation detection function is effective for a single spindle. The function cannot be executed for two or more spindles.
The spindle speed fluctuation detection function is effective for a spindle on which the currently selected position coder is mounted. Just a single position coder can be selected. Multiple position coders cannot be selected. For the selection of a position coder, see the section of "Multi spindle."
* Position coder selection signals (PC2SLC, PC3SLC, PC4SLC)
3 The parameters that become valid are the parameters of the spindle speed fluctuation detection function (No.4911, No.4912, No.4913, No.4914) for the spindle on which the currently selected position coder is mounted.

- Spindle for which to detect the spindle speed fluctuation

For the spindle for which to detect the spindle speed fluctuation, refer to the appropriate manual provided by the machine tool builder.

9.6 SPINDLE CONTROL WITH SERVO MOTOR

9.6.1 Spindle Control with Servo Motor

- Command with a program

This function provides SV speed control mode in which spindle rotation commands, S commands, are effective to a rotation axis with a servo motor and position control mode in which normal positioning can be performed. To place a servo motor under speed control, you must turn SV speed control mode ON (G96.4). Once SV speed control mode is turned ON,
S commands are effective to the servo motor until it is canceled.
You cannot specify positioning commands while SV speed control mode is ON. To perform positioning, you must cancel SV speed control mode.
To cancel SV speed control mode (turn position control mode ON), you must specify a spindle indexing command, G96.1/G96.2.
In position control mode, S commands are invalid as in the normal servo axis state. Note, however, S command information is stored; when SV speed control mode is ON, rotation starts at the already specified speed.

- Command with a signal

You can also turn SV speed control mode ON/OFF with the SV speed control mode signal. When you turn SV speed control mode ON with the signal, and if you turn SV speed control mode OFF with a program command, you must specify the SV speed control mode signal or specify G96.4 to turn SV speed control mode ON again. You can check the state of SV speed control mode with the SV speed control mode in-progress signal.

When you turn SV speed control mode signal OFF while rotating, spindle indexing is executed. Then SV speed control mode is turned off. Spindle indexing is executed with R0(absolute position 0).

Format

G96.4 P_ ; SV speed control mode ON

M03 (M04) S_ P_ ; Rotation command

S: Spindle speed [min^{-1}] (numeric value of up to five digits)
P: Spindle selection with multi-spindle control

Using G96.4 and a spindle selection command P can turn the SV speed control mode ON for each axis separately. Likewise, using a spindle indexing command (G96.1/G96.2) and a spindle selection command P can cancel the SV speed control mode, that is, turn on the position control mode ON, for each axis separately.
To specify "G96.4 P_ ;", use an independent block.
However, it is possible to specify "G96.4 P_ S_ ;".

9.6.2 Spindle Indexing Function

Format

G96.1 P_ R_ ; After spindle indexing is completed, the operation of the next block is started.

G96.2 P_ R_ ; Before spindle indexing is completed, the operation of the next block is started.

G96.3 P_ ; After it is confirmed that spindle indexing is completed, the operation of the next block is started.

P: Spindle selection with multi-spindle control
R: Stoppage angle [deg] (0 to (parameter No. 1260))

To specify address P, use parameter No. 3781 (P code for selecting a spindle in multi-spindle control).
To turn the position control mode ON without performing spindle indexing, do so after canceling the SV speed control mode by issuing a G96.1 command with no R specified when the motor is at a halt.
When the motor is rotating, issuing a G96.1 (or G96.2) command with no R specified results in the motor coming to a halt by behaving in the same manner as for R0.

NOTE
1 When using G96.2, issue G96.3 before another spindle move command, to make sure that the spindle is at a complete halt.
2 To issue G96.1, G96.2, or G96.3, use an independent block.

Spindle indexing command

- Move command

(1) Command waiting for spindle indexing to be completed
If G96.1 is issued, the next block is executed after spindle indexing is completed.
(2) Command not waiting for spindle indexing to be completed
If G96.2 is issued, the next block can be executed before spindle indexing is completed.

- Movement completion check command

G96.3 is used to check to see if spindle indexing is completed. If it has not been completed, the next block waits for spindle indexing to be completed. If it is completed, the next block is executed.

- SV speed control mode cancellation

If G96.1 is used to perform spindle indexing, the SV speed control mode is canceled when spindle indexing is completed.
If G96.2 is used to perform spindle indexing, G96.3 can be used to check to see if spindle indexing is completed and, if completed, cancel the SV speed control mode. Issuing G96.2 not followed by G96.3 cannot cancel the SV speed control mode even if spindle indexing is completed.
Start SV speed control mode cancellation at a path to which the axis of interest belongs.

- Spindle indexing command during spindle rotation

Issuing G96.1 or G96.2 with a position specified during spindle rotation causes the spindle to stop at the specified position.
Example)

```
M03 S1000 ;..... Rotation at S1000
G96.1 P1 R180. ;..... Stoppage of rotation at the 180deg position
```

- Spindle indexing speed

Issuing G96.1 or G96.2 causes a move speed to be dedicated to spindle indexing. Specify the move speed for spindle indexing, using parameter No. 11012.

Spindle indexing command (absolute coordinate/machine coordinate)

Using bit 0 (SIC) of parameter No. 11005 can select which coordinate system, absolute or machine, is to be used in spindle indexing.

Example:

If the difference between the machine and absolute coordinates (machine coordinate – absolute coordinate) is 100.000:

- Spindle indexing performed by turning the SV speed control mode signals SRVON1 to SRVON8 OFF (“1” to “0”) or issuing a spindle indexing G code (with no R specified)
  1. If bit 0 (SIC) of parameter No. 11005 = 0:
     Spindle indexing is performed with a machine coordinate of 100.000 and an absolute coordinate of 0.000.
  2. If bit 0 (SIC) of parameter No. 11005 = 1:
     Spindle indexing is performed with a machine coordinate of 0.000 and an absolute coordinate of 260.000.
- G code-based spindle indexing (with R specified)
  Assuming that spindle indexing is specified with R100.000:
  1. If bit 0 (SIC) of parameter No. 11005 = 0:
     Spindle indexing is performed with a machine coordinate of 200.000 and an absolute coordinate of 100.000.
  2. If bit 0 (SIC) of parameter No. 11005 = 1:
     Spindle indexing is performed with a machine coordinate of 100.000 and an absolute coordinate of 0.000.

10 TOOL FUNCTION (T FUNCTION)

Chapter 10, "TOOL FUNCTION (T FUNCTION)", consists of the following sections:

10.1 TOOL SELECTION FUNCTION 202
10.2 TOOL MANAGEMENT FUNCTION 203
10.3 TOOL MANAGEMENT EXTENSION FUNCTION 217
10.4 TOOL MANAGEMENT FUNCTION FOR OVERSIZE TOOLS 226
10.5 TOOL LIFE MANAGEMENT 228

10.1 TOOL SELECTION FUNCTION

By specifying an up to 8-digit numerical value following address T, a code signal and a strobe signal are transmitted to the machine tool. This is used to select tools on the machine.
One T code can be commanded in a block. Refer to the machine tool builder's manual for the number of digits commendable with address T and the correspondence between the T codes and machine operations.
When a move command and a T code are specified in the same block, the commands are executed in one of the following two ways:

(i) Simultaneous execution of the move command and T function commands.

(ii) Executing T function commands upon completion of move command execution.

The selection of either (i) or (ii) depends on the machine tool builder's specifications. Refer to the manual issued by the machine tool builder for details.

Explanations

T

The value after the T code indicates the desired tool. Part of the value is also used as a tool offset number that specifies the amount of tool offset or the like. The tool can be selected as follows according to the specification method and parameter settings.

Description of a T code (Note 1)		How to specify the offset number for each parameter setting (Note 2)
Bit 1 (LGN) of parameter No. 5002 = 0	**Bit 1 (LGN) of parameter No. 5002 = 1**	
TOOOOOOO O ↑ ↑ Tool selection Tool geometry tool wear offset	TOOOOOOO O ↑ ↑ Tool selection Tool wear tool geometry offset offset	The tool wear offset number is specified using the low-order one digit of a T code. When parameter No.5028 is set to 1
TOOOOOO OO ↑ ↑ Tool selection Tool geometry tool wear offset	TOOOOOO OO ↑ ↑ Tool selection Tool wear tool geometry offset offset	The tool wear offset number is specified using the low-order two digits of a T code. When parameter No.5028 is set to 2
TOOOOO OOO ↑ ↑ Tool selection Tool geometry tool wear offset	TOOOOO OOO ↑ ↑ Tool selection Tool wear tool geometry offset offset	The tool wear offset number is specified using the low-order three digits of a T code. When parameter No.5028 is set to 3

NOTE
1 The maximum number of digits of a T code can be specified by parameter No.3032 as 1 to 8.
2 When parameter No.5028 is set to 0, the number of digits used to specify the offset number in a T code depends on the number of tool offsets.
Example)
When the number of tool offsets is 1 to 9: Low-order one digit
When the number of tool offsets is 10 to 99: Low-order two digits
When the number of tool offsets is 100 to 999: Low-order three digits

Refer to the machine tool builder's manual for correspondence between the T-code and the tool and the number of digit to specify tool selection.

Example (T2+2)
N1G00X1000Z1400;
N2T0313; (Select tool No. 3 and tool offset value No. 13)
N3X400Z1050;

10.2 TOOL MANAGEMENT FUNCTION

Overview

The tool management function totally manages tool information including information about tool offset and tool life.

Explanation

A tool type number is specified with a T code. The tool type number is any number the user can define freely. With tool type numbers, tools can be grouped by various conditions such as life, compensation value, and cutting conditions. When each type is assumed to have a single tool, tool type numbers are equivalent to unique tool numbers.

For each tool, an information storage area is prepared in the CNC (SRAM). This area contains information such as a tool type number, tool life, tool status (such as a breakage condition), tool compensation number (H, D, G, or W), spindle speed (S), cutting feedrate (F), and freely-definable customize data. Such data is called tool management data.
A cartridge management table that links cartridge information and tool management data is provided so that the CNC can manage the cartridges of the machine and tool change operations. In addition, areas for managing the tools in the spindle and tool standby positions are prepared.

When a tool type number is specified using a T code command, a tool that has the tool type number and the shortest life is searched for, and the cartridge number and pot number of the location where the tool is stored are output to the PMC. Then, a tool change operation using the cartridge number and pot number is enabled by the PMC ladder sequence.

Tool information in the CNC is managed by the tool management data and cartridge management table (including the spindle management table and standby position table).

- Tool management data

As tool management data, information about each tool is registered with a tool management data number. The following number of sets of tool management data can be used:

Tool management function 64 sets	64 sets in total
Tool management function 240 sets	240 sets in total
Tool management function 1000 sets	1000 sets in total

NOTE
For the number of tool management data sets, refer to the relevant manual issued by the machine tool builder.

- Details of data

The following details the tool management data registered for each data number:

- Tool type number (T code)

Item	Description
Data length	4byte
Valid data range	0,1 to 99,999,999

- Tool life counter

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds
Valid data range	When the number of use times is specified: 0 to 99,999,999 times When time is specified: 0 to 3,599,999 s (999 hours 59 minutes 59 seconds)

The value of an increment counter, that is, the number of use times (time) is indicated. The remaining life value is [the maximum tool life value minus tool life counter value].

- Maximum tool life value

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds
Valid data range	When the number of use times is specified: 0 to 99,999,999 times When time is specified: 0 to 3,599,999 s (999 hours 59 minutes 59 seconds)

- Notice life value

Item	Description
Data length	4byte
Unit of data	When the number of use times is specified: Times When time is specified: Seconds
Valid data range	When the number of use times is specified: 0 to 99,999,999 times When time is specified: 0 to 3,599,999 s (999 hours 59 minutes 59 seconds)

If a value other than 0 is set, the tool life expiration notice signal TLCHB or TLCHBx is output when the remaining life value of the tool (= maximum tool life value minus tool life counter value) has reached the set value.

- Tool life status

Item	Description
Data length	1byte
Detail data	0: Life management is not performed. 1: Tool not yet used 2: Life remains. 3: Life expired. 4: Tool breakage (skip)

The machine (PMC) determines tool breakage and stores corresponding information through the window. In tool management of the CNC, a broken tool is regarded as being equivalent to tools whose lives have expired.

- Tool information

Item	Description
Data length	1 byte (flag data)
#0 RGS	0: Tool management data is invalid. (-) 1: Tool management data is valid. (R)
#1 TIM	0: Type for counting the number of use times (C) 1: Type for counting time (T)
#2 BDT	0: Normal tool (N) 1: Oversize tool (B)
#3 LOC	0: Data accessible (U) 1: Data not accessible (L)
#4 SEN	When the tool life status indicates that tool management is not performed: 0: This tool is not searched. (-). 1: This tool is searched. (S)
#5 to #7	Reserved

When RGS is set to 0 in tool management data, the tool management data is regarded as not being registered even when values are set for other items.

NOTE
Make sure you specify the same life count type for all tools of the same type. You can check tools of the same type for any difference in life count type, using the "check function."

M

- Tool length compensation number (H)

Item	Description
Data length	2byte
Valid data range	0 to 999

- Cutter compensation number (D)

Item	Description
Data length	2byte
Valid data range	0 to 999

T

- Tool geometry compensation number (G)

Item	Description
Data length	2byte
Valid data range	0 to 999

- Tool wear compensation number (W)

Item	Description
Data length	2byte
Valid data range	0 to 999

NOTE
When the machine control type is the combined system type, tool length compensation and cutter compensation numbers are used for paths for the machining center system, and for paths for the lathe system, tool geometry compensation and tool wear compensation numbers are used.

- Spindle speed (S)

Item	Description
Data length	4byte
Unit of data	min^{-1}
Valid data range	1 to 99,999

- Feedrate (F)

Item	Description
Data length	4byte
Unit of data	mm/min, inch/min, deg/min, mm/rev, inch/rev
Valid data range	0 to 99,999,999

As additional tool management data, areas for setting customize data (5 data items including customize data 0 and customize data 1 to 4) are provided without defining specific usage. The user can use these customize data areas freely to set a warning life value, cutting resistance, override value, spindle current value, maximum and minimum S/F, and other items according to the targeted application.

- Customize data 0

Item	Description
Data length	1 byte (bit type)
Valid data range	0 or 1 on a bit-by-bit basis

- Customize data 1 to 4 (to 20) (to 40)

Item	Description
Data length	4byte
Valid data range	-99,999,999 to 99,999,999

NOTE
For the maximum number of tool management function customization data items, refer to the relevant manual issued by the machine tool builder.

- Cartridge management table

The storage status of tools in cartridges is managed with the cartridge management table.

- Multiple cartridge numbers can be defined. (Numbers from 1 to 4, up to four cartridges)
- The maximum number of pots for all cartridges is 64, 240, or 1000, which depends on whether a tool management data option is selected.
- Cartridge numbers and pot numbers can be assigned freely within cartridge management data by parameter setting (described later).
- The tool management data number adjacent to a pot number is linked with tool data defined in the tool management table. Therefore, the tool attached to the pot is indicated.
- Zero set as a data number indicates that no tool is attached.
- The cartridge management table can be read from and written to through the PMC window and FOCAS2.
- The spindle management table and standby position table are provided to indicate special cartridge positions.

- Spindle positions and standby positions, regarded as special cartridge positions, have fixed cartridge numbers 11 to 14 (the positions of the first to fourth spindles) and 21 to 24 (the first to fourth standby positions).
- With the PMC window, the spindle position table and standby position table can be read from and written to.
- Tool life counting is performed only for the tools at the spindle positions.

- Multi-path system

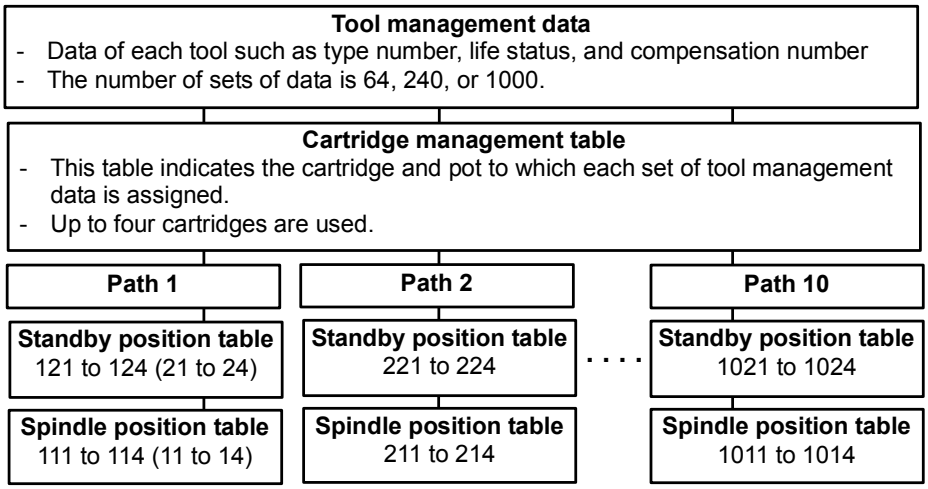
The tool management data and cartridge management table are common data among the paths. The spindle management table and standby position table, however, are treated as independent data for each path.

When the spindle table or standby position table is specified as a cartridge from the PMC window, specify the following, in which the path number is set in the hundred's place:

	Spindle position			
	First	**Second**	**Third**	**Fourth**
First path	111(11)	112(12)	113(13)	114(14)
Second path	211	212	213	214
Third path	311	312	313	314
Fourth path	411	412	413	414
Fifth path	511	512	513	514
Sixth path	611	612	613	614
Seventh path	711	712	713	714
Eighth path	811	812	813	814
Ninth path	911	912	913	914
Tenth path	1011	1012	1013	1014

	Standby position			
	First	**Second**	**Third**	**Fourth**
First path	121(21)	122(22)	123(23)	124(24)
Second path	221	222	223	224
Third path	321	322	323	324
Fourth path	421	422	423	424
Fifth path	521	522	523	524
Sixth path	621	622	623	624
Seventh path	721	722	723	724
Eighth path	821	822	823	824
Ninth path	921	922	923	924
Tenth path	1021	1022	1023	1024

NOTE
When specifying 111, 121, and so on to specify the first path, you may specify just 11, 21, and so on.



- M/T series

The tool management data and cartridge management table are data common to the M and T series.

- Tool life management and tool change

The CNC performs tool life management, regarding tools having the same tool type number as a group. When a tool type number (T code) is specified by an NC program, the tool management data registered in the CNC is searched to find a tool having the shortest life among the tools with the same tool type number.
The cartridge number and pot number corresponding to the searched tool are output as a T code signal to the PMC. Based on the output cartridge number and pot number, the PMC performs preparation for a tool change (to the next tool). A different tool can also be selected on the PMC side.
The CNC performs tool life counting for each tool that is at a spindle position in the spindle management table.

When the lives of all tools having the tool type number specified by the T code have expired, alarm PS5317, “ALL TOOL LIFE IS OVER” is issued. If there is a tool in the spindle position or standby position, that tool is selected to continue machining.
As the T code signal, a specified tool type number instead of the cartridge number and pot number can also be output directly by parameter setting.

There are two types of tool life management counting methods: counting the number of use times and counting cutting time. One of the counting methods is set in tool information of tool management data.

Other major specifications related to tool life management are as follows:
Tool type number (T code) : Up to 8 digits (1 to 99,999,999)
Maximum tool life value : 99,999,999 times when the number of use times is specified
: 999 hours 59 minutes 59 seconds when time is specified
Life count interval when time is specified : 1 second
Tool life management count restart M code : Enabled
Tool life count override : Enabled

To specify a certain tool directly without performing tool life management by tool type number, use the following format:

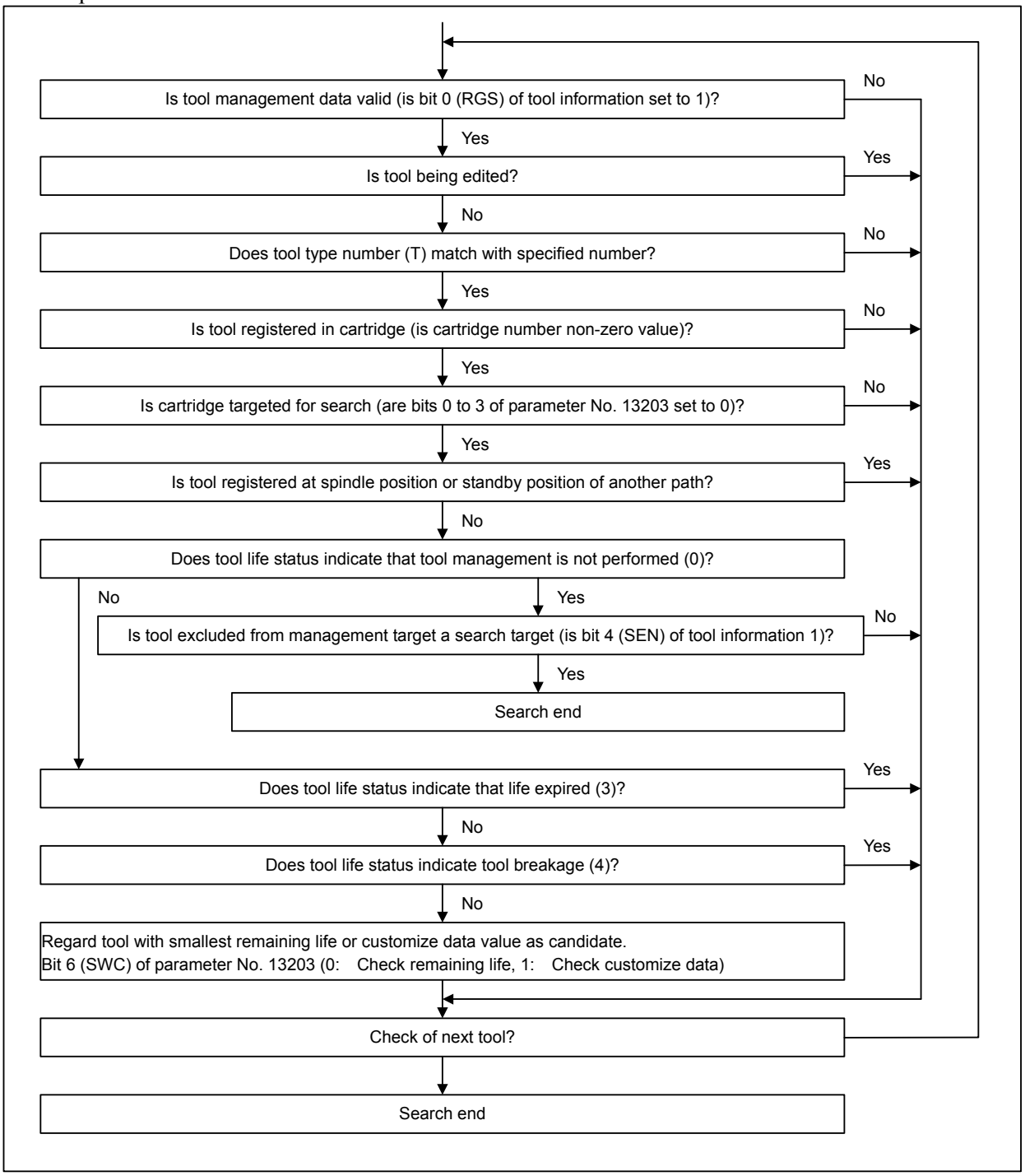
M (value in parameter No. 13252) T (cartridge number) (pot number) ;

When parameter No. 13252 is set to 333, the cartridge number is 2, and the pot number is 27, for example, the following command is specified:

M333 T20027;

- Tool search order

Tools having a tool type number (T) specified by a program are searched sequentially from tool management data number 1 while registered data contents are checked. The following shows how a search operation is made within the NC:



- System variables

The following tool management data of the tool being used as a spindle after a tool change by M06 and the tool to be used next which is specified by a T code can be read through custom macro variables:

Being used	Item	
#8401	Tool management data number	
#8402	Tool type number (T)	
#8403	Tool life counter	
#8404	Maximum tool life value	
#8405	Tool notice life value	
#8406	Tool life status	
#8407	Customize data 0 (bit)	
#8408	Tool information	
#8409	Tool length compensation number (H)	For machining center systems
#8410	Cutter compensation number (D)	
#8411	Spindle speed (S)	
#8412	Cutting feedrate (F)	
#8413	Tool geometry compensation number (G)	For lathe systems
#8414	Tool wear compensation number (W)	
#8431	Customize data 1	
#8432	Customize data 2	
#8433	Customize data 3	
#8434	Customize data 4	
#8435	Customize data 5	
#8436	Customize data 6	
#8437	Customize data 7	
#8438	Customize data 8	
#8439	Customize data 9	
#8440	Customize data 10	
#8441	Customize data 11	
#8442	Customize data 12	
#8443	Customize data 13	
#8444	Customize data 14	
#8445	Customize data 15	
#8446	Customize data 16	
#8447	Customize data 17	
#8448	Customize data 18	
#8449	Customize data 19	
#8450	Customize data 20	
#8451	Customize data 21	
#8452	Customize data 22	
#8453	Customize data 23	
#8454	Customize data 24	
#8455	Customize data 25	
#8456	Customize data 26	
#8457	Customize data 27	
#8458	Customize data 28	
#8459	Customize data 29	
#8460	Customize data 30	
#8461	Customize data 31	
#8462	Customize data 32	
#8463	Customize data 33	
#8464	Customize data 34	
#8465	Customize data 35	
#8466	Customize data 36	
#8467	Customize data 37	

Being used	Item
#8468	Customize data 38
#8469	Customize data 39
#8470	Customize data 40

When a cartridge number of a spindle position (11 to 14) or standby position (21 to 24) is specified in #8400, information about the corresponding position can be read.
If the spindle position table or standby position table has an empty pot, <empty> is read from #8402 to #8470.
Value 0 is read from #8401 (data number).

Therefore, machining conditions registered in tool management data can be specified directly by coding, for example, D#8410, H#8409, S#8411, and F#8412 with a tool change macro (such as M06). Similarly, customize data can be referenced by a custom macro, and machining programs can be customized according to the tool used.

NOTE
For the maximum number of tool management function customization data items, refer to the relevant manual issued by the machine tool builder.

- Specifying a tool compensation number

M

When parameter No. 13265 is 0, a compensation number registered as tool management data of a tool attached at a spindle position can be selected by specifying H99 or D99.
(99 is treated as a special number, so 99 cannot be specified directly as a compensation number.)
When other than 0 is set in parameter No. 13265, the number set in the parameter can be specified instead of 99. For example, if parameter No. 13265 is 3, specifying H3 specifies the tool length compensation number registered for the tool at the spindle position.

T

When the tool management function is not used, a tool compensation number is specified also with a T code; when the tool management function is used, the T code is used only to specify a tool type number, so a tool compensation number must be specified with address D.
Except the specifications for addresses, the specifications for the number of permissible digits (parameter No. 3032) and the number of digits consisting of a compensation number (parameter No. 5028), separation of geometry compensation numbers and wear compensation numbers (bit 1 (LGN) of parameter No. 5002), and so on are the same as for T.

If parameter No. 13265 is set to 0, when a compensation number registered for a tool attached at a spindle position is specified, the specification format varies according to the number of digits as follows, which is the same as for conventional T:
When the compensation number is 1 digit long : D9
When the compensation number is 2 digits long : D99
When the compensation number is 3 digits long : D999
Because 9, 99, or 999 is treated as a special number, it cannot be specified directly as a compensation number.
Unless parameter No. 13265 is set to 0, the number set in the parameter can be specified instead of 9, 99, or 999. When parameter No. 13265 is set to 3, specifying D3 specifies the tool geometry compensation number and tool wear compensation number registered for a tool attached at the spindle position.

Multi-path system

Depending on whether the local path is a machining center system or a lathe system, tool compensation numbers are specified by using one of the above methods.

Spindle selection

When specifying compensation numbers of a tool attached to a spindle other than the first spindle, specify the spindle number with address P within the same block that contains H/D. When specifying the first spindle, you can omit P.

D99 P3; Specifying compensation numbers registered for the tool attached at the third spindle
D99; Specifying compensation numbers registered for the tool attached at the first spindle

- Read/write operations for tool management data and cartridge management table

The tool management data and cartridge management table can be read from and written to by using the CNC/PMC data window library (FOCAS2). Therefore, a specific tool management system including all available tool data not registered in the CNC can be built easily by using the personal computer function.

Similarly, the tool management data and cartridge management table can also be read from and written to using the PMC window.

Tool management data of the tool being used as a spindle after a tool change operation performed by M06 and the tool being selected by specifying a T code can be read using a custom macro.

The user can modify tool management data by MDI from the tool management function screen of the CNC. Addition, modification, and deletion of the above tool management data can be made from a part program (G10). Such data can also be input to and output from external I/O equipment by using the ALL I/O screen and tool management function screen.

- G10 format

Addition, modification, and deletion are performed for the tool management data and cartridge management table from programs. For use of this function, the programmable data input option function is required.

If a format error is found in the commands from G10 L75/L76/L77 to G11, or if a value beyond the valid data range is specified, alarm PS5312, “ILLEGAL COMMAND IN G10 L75/76/77” is issued. In such a case, correct the program. Within the range from G10 to G11, no decimal point can be specified with any address. If a decimal point is specified, alarm PS0007, “ILLEGAL USE OF DECIMAL POINT” results.

- Registering new tool management data

Tool management data can be registered. When data is output to an external device from the tool management data screen, this format is used. The specification of those items that are not registered may be omitted.

G10 L75 P1;
N_ ; Tool management data number specification
T_ C_ L_ I_ B_ Q_ H_ D_ S_ F_ J_ K_ ;
P0 R_ ; Customization data 0
P1 R_ ; Customization data 1
P2 R_ ; Customization data 2
P3 R_ ; Customization data 3
P4 R_ ; Customization data 4
N_ ; Next tool management data number specification
:
G11;

N_	Tool management data No.	1 to 64 (1 to 240, 1 to 1000)
T_	Tool type No. (T)	0 to 99,999,999
C_	Tool life counter	0 to 99,999,999
L_	Maximum tool life	0 to 99,999,999
I_	Noticed life	0 to 99,999,999
B_	Tool life state	0 to 4
Q_	Tool information	Bit format (8 bits)
H_	Tool length compensation No. (H)	0 to 999 (M series)
D_	Cutter compensation No. (D)	0 to 999 (M series)
S_	Spindle speed (S)	0 to 99,999
F_	Cutting feedrate (F)	0 to 99,999,999
J_	Tool geometry compensation No. (G)	0 to 999 (T series)
K_	Tool geometry compensation No. (W)	0 to 999 (T series)
P_	Customization data No.	0 to 4 (0 to 20, 0 to 40)
R_	Customization data value	-99,999,999 to 99,999,999

Specify customization data in the following format:

P (customization-number) R (value)

Use the bit format only when specifying the customization data 0 (P0). Specify other data in the binary format. The specification of customization data that need not be set may be omitted.

Example)

```
G10 L75 P1;
N1 ;                        Tool management data No.1
T10000001 C0 L1000 B1 Q00000001 H1 D1 S4000 F10000 ;
P0 R11101101 ;              Customization data 0
P4 R99999999 ;              Customization data 4
N2 ;                        Tool management data No.2
T10000002 C0   L2000 B1 Q00000001 H2 D2 S3000 F6000 ;
P1 R200 ;                   Customization data 1
P2 R10 ;                    Customization data 2
N5 ;                        Tool management data No.5
T10000005 C0   L5000 B1 Q00000001 H5 D5 S5000 F11000 ;
P2 R1 ;                     Customization data 2
P3 R500 ;                   Customization data 3
G11 ;
```

Registering tool management data No.1
Registering tool management data No.2
Registering tool management data No.5

Modifying tool management data

Tool management data can be modified.
The specification of those items that are not modified may be omitted.

```
G10 L75 P2 ;
N_ ;
T_ C_ L_ I_ B_ Q_ H_ D_ S_ F_ J_ K_ ;
P_ R_ ;
N_ ;
 :
G11 ;
```

Deleting tool management data

The data of a specified data number can be deleted from tool management data.
The cartridge management table data corresponding to a deleted tool management data number is also deleted. (The tool management data number in the cartridge management table is cleared to 0.)

```
G10 L75 P3 ;
N_ ;
N_ ;
 :
N_ ;
G11 ;
```

Registering new cartridge management table data

A tool management data number can be registered with a free pot in the cartridge management table.

```
G10 L76 P1 ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
G11 ;
```

For a spindle position table and standby position table, only cartridge number data is specified.
Example)

```
G10 L76 P2 ;
N11 R1;   Changes the tool management data number of the spindle position to No. 1.
N21 R29; Changes the tool management data number of the standby position to No. 29.
G11 ;
```

Modifying the cartridge management table

Tool management data numbers in the cartridge management table can be modified.

```
G10 L76 P2 ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
G11 ;
```

For a spindle position table and standby position table, only cartridge number data is specified.
Example)

G10 L76 P2 ;
N11 R1; Changes the tool management data number of the spindle position to No. 1.
N21 R29; Changes the tool management data number of the standby position to No. 29.
G11 ;

Deleting cartridge management table data

Tool management data numbers can be deleted from the cartridge management table.

```
G10 L76 P3 ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
N cartridge-number P pot-number R tool-management-data-number ;
G11 ;
```

For a spindle position table and standby position table, only cartridge number data is specified.
Example)

G10 L76 P3 ;
N11 ; Deletes the tool management data number of the spindle position (clears the number to 0).
N21 ; Deletes the tool management data number of the standby position (clears the number to 0).
G11 ;

Naming customization data

The display name of customization data (0 to 40) can be set.

```
G10 L77 P1 ;
N_ ;
P_ R_ ;
P_ R_ ;
;
N_ ;
P_ R_ ;
P_ R_ ;
G11 ;
```

N_: Customization data No. (0 to 40)
P_: Character No. (1 to 16)
R_: Character code (ANK or shifted JIS)
- When a shifted JIS code is used, an area for two characters is used.
- Specify 0 to clear data.

- A set display name can be checked on the tool management data screen only.
- When no name is registered, a name such as the default "Customize 3" is displayed.
- A name consisting of up to 16 characters is displayed. For a name shorter than 16 characters, register 0 in the empty area. Those characters that are registered immediately before 0 are displayed.
- To clear data, set 0 as a character code.
- No character code data check is made.

When the name of customization data 3 is set as "MEASURE5", for example, specify the following:
Example)

G10 L77 P1 ;

```
N3 ;        Specifies customization data 3.
P1 R32 ;    character ‘space’ ASCII code 20h
P2 R77 ;    character “M” ASCII code 4Dh
P3 R69 ;    character “E” ASCII code 45h
P4 R65 ;    character “A” ASCII code 41h
P5 R83 ;    character “S” ASCII code 53h
P6 R85 ;    character “U” ASCII code 55h
P7 R82 ;    character “R” ASCII code 54h
P8 R69 ;    character “E” ASCII code 45h
P9 R53 ;    character “5” ASCII code 35h
P10 R0 ;    Clears data. (Not displayed. End)
G11 ;
```

Naming tool life states

The display name of a tool life state (0 to 4) can be set.

```
G10 L77 P2 ;
N_ ;
P_ R_ ;
P_ R_ ;
N_ ;
P_ R_ ;
P_ R_ ;
G11 ;
```

N_: Tool life state (0 to 4)
P_: Character No. (1 to 12)
R_: Character code (ANK or shifted JIS)
- When a shifted JIS code is used, an area for two characters is used.
- Specify 0 to clear data.

- A set display name can be checked on the tool management data screen only.
- If no name is registered, a default such as "Remaining" and "Unremaining" is displayed.
- A name consisting of up to 12 characters is displayed. For a name shorter than 12 characters, register 0 in the empty area. Those characters that are registered immediately before 0 are displayed.
- To clear data, set 0 as a character code.
- No character code data check is made.
- The defaults are as follows:
  - 0: Invalid
  - 1: Remaining
  - 2: Remaining
  - 3: Unremaining
  - 4: Broken

To set "NORMAL" as the name of tool life state 2 (Remaining), specify the following:
Example)

```
G10 L77 P2 ;
N2 ;        Specifies tool life state 2.
P1 R78 ;    character “N” ASCII code 4Eh
P2 R79 ;    character “O” ASCII code 4Fh
P3 R82 ;    character “R” ASCII code 52h
P4 R77 ;    character “M” ASCII code 4Dh
P5 R65 ;    character “A” ASCII code 41h
```

```
P6 R76 ;        character “L” ASCII code 4Ch
P7 R0 ;         Clears data. (Not displayed. End)
G11 ;
```

10.3 TOOL MANAGEMENT EXTENSION FUNCTION

Overview

The following functions have been added to the tool management function:

1. Customization of tool management data display
2. Setting of spindle position/standby position display
3. Input of customize data with the decimal point
4. Protection of various tool information items with the KEY signal
5. Selection of a tool life count period
6. Each tool data screen
7. Total life time display for tools of the same type

They will be explained in detail below.

10.3.1 Customization of Tool Management Data Display

With the tool management data screen display customization function, the display positions of screen elements (type number, tool information, life counter, and so forth) on the tool management screen can be changed and whether to display or hide such screen elements can be chosen using the G10 format. This function enables a customized tool management screen to be configured.

Format

G10L77P3;

N_ R_;

G11;

N_: Tool management data screen display position number
R_: Tool management data display item

Explanation

- number (N_)

- A display position number represents an ordinal number from the leftmost position on the tool management data screen.
- As N, a number from 1 to 200 can be specified.

- Tool management data display item (R_)

- For a value to be set (in R_), see the table below.
- For actual setting, consider a display width plus 1.
- Items common to machining center systems and lathe systems

R	Item	Display width	Remarks
-1	End of setting	-	
0	Blank column	10	
1	No.	4	For spindle/standby positions for 10 paths
2	TYPE-NO.	8	
3	MG	4	
4	POT	5	
5	T-INFORMATION	10	

R	Item	Display width	Remarks
6	L-COUNT	10	
7	MAX-LIFE	10	
8	NOTICE-L	10	
9	L-STATE	6 or 12	The display width is switched by bit 1 of parameter No. 13201.
10	S (Spindle speed)	10	
11	F (Feedrate)	10	
12	Tool figure number (A)	3	

- Offset-related items for machining center systems

R	Item	Display width	Remarks
20	H (Tool length compensation number)	4	
21	D (Cutter compensation number)	4	
22	OFFSET-M	10	Tool offset memory A
23	GEOMETRY-M	10	Tool offset memory B
24	WEAR-M	10	
25	GEOMETRY(H)	10	Tool offset memory C
26	WEAR(H)	10	
27	GEOMETRY(D)	10	
28	WEAR(D)	10	

- Offset-related items for lathe systems

R	Item	Display width	Remarks
40	TG (Tool geometry compensation number)	4	
41	TW (Tool wear compensation number)	4	
42	OFFSET-X	10	
43	OFFSET-Z	10	
44	OFFSET-R	10	Tool nose radius compensation
45	TIP	10	
46	OFFSET-Y	10	Y-axis offset
47	OFFSET-B	10	B-axis control
48	GEOMETRY (X)	10	Tool geometry/wear compensation
49	WEAR (X)	10	
50	GEOMETRY (Z)	10	
51	WEAR (Z)	10	
52	GEOMETRY (R)	10	Tool nose radius compensation, Tool geometry/wear compensation
53	GEOMETRY TIP	10	
54	WEAR (R)	10	
55	WEAR TIP	10	
56	GEOMETRY (Y)	10	Y-axis offset, Tool geometry/ wear compensation
57	WEAR (Y)	10	
58	GEOMETRY (B)	10	B-axis control, Tool geometry/ wear compensation
59	WEAR (B)	10	
60	GEOMETRY (X2)	10	2nd geometry tool offset
61	GEOMETRY (Z2)	10	
62	GEOMETRY (Y2)	10	

Items related to customize data

R	Item	Display width	Remarks
80	CUSTOM 0	10	
81	CUSTOM 1	10	
82	CUSTOM 2	10	
83	CUSTOM 3	10	

R	Item	Display width	Remarks
84	CUSTOM 4	10	
85	CUSTOM 5	10	Tool management function customize data extension (5 to 20) or tool management function customize data extension (5 to 40)
86	CUSTOM 6	10	
87	CUSTOM 7	10	
88	CUSTOM 8	10	
89	CUSTOM 9	10	
90	CUSTOM 10	10	
91	CUSTOM 11	10	
92	CUSTOM 12	10	
93	CUSTOM 13	10	
94	CUSTOM 14	10	
95	CUSTOM 15	10	
96	CUSTOM 16	10	
97	CUSTOM 17	10	
98	CUSTOM 18	10	
99	CUSTOM 19	10	
100	CUSTOM 20	10	
101	CUSTOM 21	10	Tool management function customize data extension (5 to 40)
102	CUSTOM 22	10	
103	CUSTOM 23	10	
104	CUSTOM 24	10	
105	CUSTOM 25	10	
106	CUSTOM 26	10	
107	CUSTOM 27	10	
108	CUSTOM 28	10	
109	CUSTOM 29	10	
110	CUSTOM 30	10	
111	CUSTOM 31	10	
112	CUSTOM 32	10	
113	CUSTOM 33	10	
114	CUSTOM 34	10	
115	CUSTOM 35	10	
116	CUSTOM 36	10	
117	CUSTOM 37	10	
118	CUSTOM 38	10	
119	CUSTOM 39	10	
120	CUSTOM 40	10	

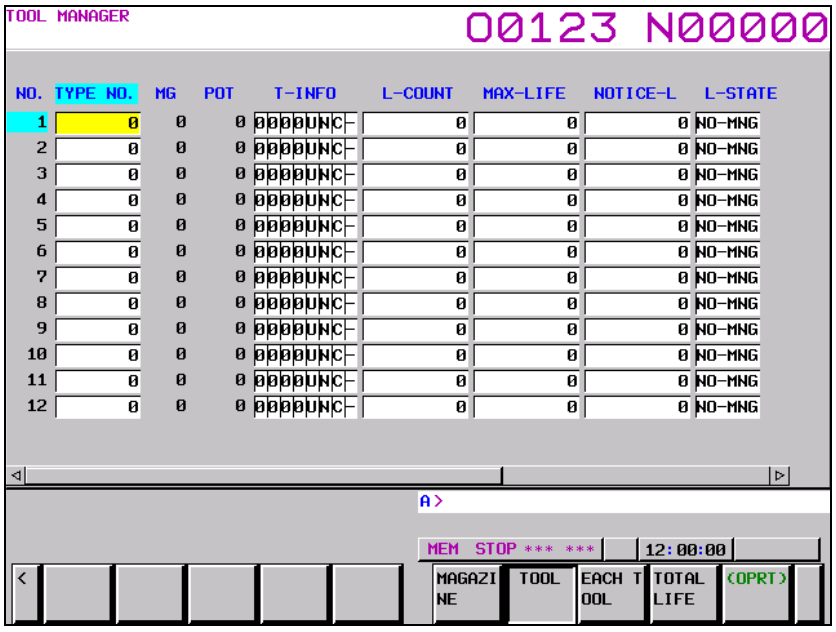
NOTE
1 If G10 L77 P3 is terminated normally, the power must be turned off before operation is continued.
2 The setting becomes effective after the power is turned off then back on.

Example

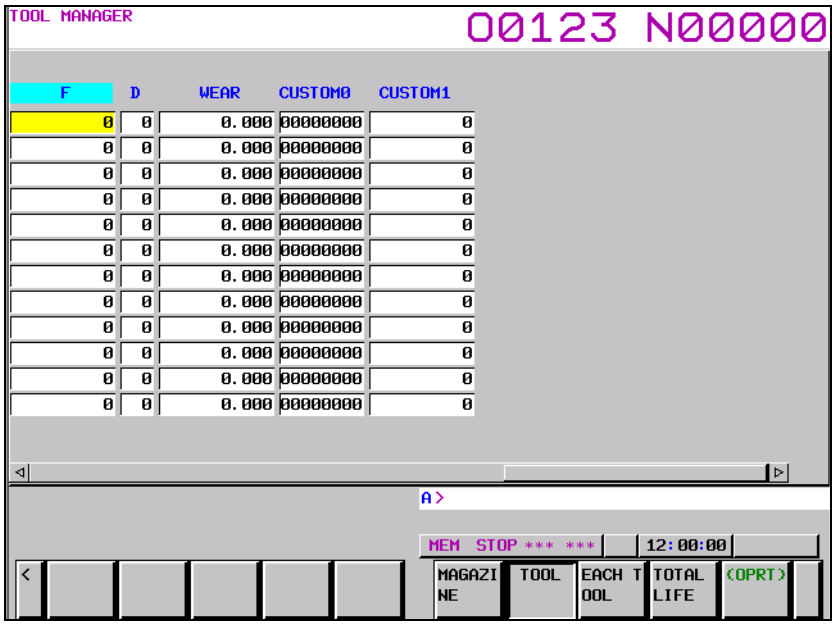
Example of setting tool offset memory A

G10L77P3;	Set tool management data screen display customization
N1 R1;	Set No. as number 1
N2 R2;	Set TYPE-NO. as number 2
N3 R3;	Set MG as number 3
N4 R4;	Set POT as number 4
N5 R5;	Set T-INFORMATION as number 5
N6 R6;	Set L-COUNT as number 6
N7 R7;	Set MAX-LIFE as number 7
N8 R8;	Set NOTICE-L as number 8
N9 R9;	Set L-STATE as number 9
N10 R11;	Set F (Feedrate) as number 10
N11 R21;	Set D as number 11
N12 R22;	Set OFFSET-M as number 12
N13 R80;	Set CUSTOM 0 as number 13
N14 R81;	Set CUSTOM 1 as number 14
N15 R-1;	End
G11;	Cancel the setting mode

On the tool management data screen, the first page displays "No., Type Number, MG, Pot, Tool Information, Life Counter, Maximum Life, Noticed Life, and Life State".
The second page displays "F (Feedrate), D (Cutter Compensation Number), Tool Offset-M, Customize Data 0, and Customize Data 1".



Example 1: Page 1



Example 1: Page 2

NOTE
1 This setting is enabled when bit 0 (TDC) of parameter No. 13201 is set to 1.
2 Up to 20 pages can be set.
3 Be sure to specify an end.
4 If an item that requires the corresponding option is specified without specifying the option, the item is displayed as a blank field.
5 No items after an end are displayed.
(Example) In the case of N1 → R1 (No.), N2 → R2 (Type Number), N 3 → R3 (MG), N4 → R-1 (End), N5 → R4 (Pot), only the items "No., Type Number, and MG" are displayed on the screen.
6 If G10 L77 P3 is specified, the setting immediately before is initialized.
7 If an invalid value is set in R (tool management data display item), a blank field is displayed.

10.3.2 Setting of Spindle Position / Standby Position Display

In MG on the tool management data screen, a spindle position or standby position is displayed as a number such as 11, 12, and 13. With the spindle position/standby position display setting function, three arbitrary characters can be displayed using the G10 format.

Format

G10L77P4;
N_;
P_R_;
G11;

N_: Spindle position/standby position setting
P_: Character number
R_: Character code

Explanation

- Spindle position/standby position setting (N_)

Specify a spindle position or standby position to be renamed. The table below indicates the values to be specified.

	Spindle position			
	First	Second	Third	Fourth
1st path	111	112	113	114
2nd path	211	212	213	214
3rd path	311	312	313	314
4th path	411	412	413	414
5th path	511	512	513	514
6th path	611	612	613	614
7th path	711	712	713	714
8th path	811	812	813	814
9th path	911	912	913	914
10th path	1011	1012	1013	1014

	Standby position			
	First	Second	Third	Fourth
1st path	121	122	123	124
2nd path	221	222	223	224
3rd path	321	322	323	324
4th path	421	422	423	424
5th path	521	522	523	524
6th path	621	622	623	624
7th path	721	722	723	724
8th path	821	822	823	824
9th path	921	922	923	924
10th path	1021	1022	1023	1024

- Character number (P_)

Specify a character number (1 to 3). Up to three characters are displayed. If a character string to be specified is shorter than three characters, specify 0 in the leading blank character position(s). A character string immediately before 0 is displayed.

- Character code (R_)

Set the name of a spindle position/standby position by using a character code (ASCII code or Shift JIS code).

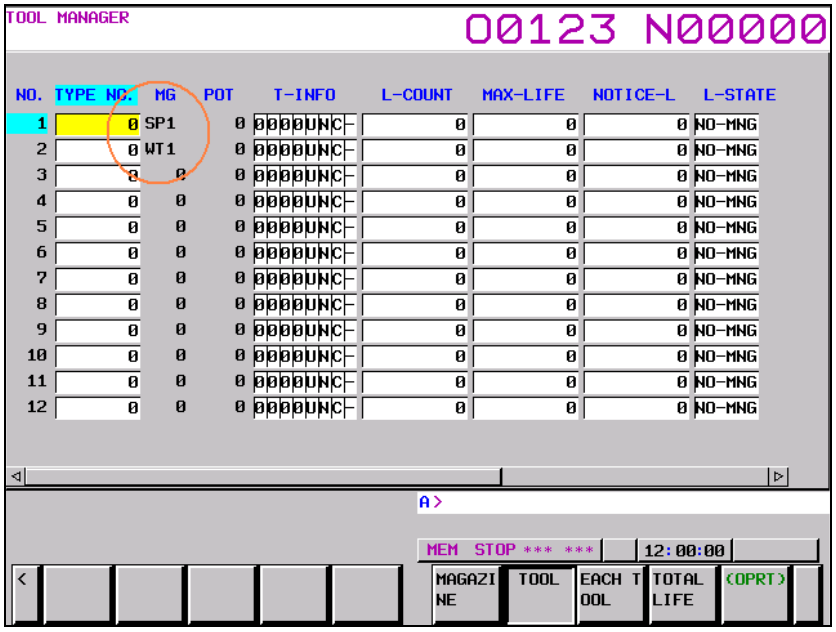
- For a character code, use ASCII code or Shift JIS code.
- No character code data check is made.

Example

When spindle 1 is named "SP1", and standby 1 is named "WT1"

G10L77P4;	Set spindle position/standby position
N111;	Specify spindle 1
P1 R83;	53h as ASCII code for "S"
P2 R80;	50h as ASCII code for "P"
P3 R49;	31h as ASCII code for "1"
N121;	Specify standby 1
P1 R87;	57h as ASCII code for "W"
P2 R84;	54h as ASCII code for "T"
P3 R49;	31h as ASCII code for "1"
G11;	Cancel the setting mode

In the MG item on the tool management data screen, spindle 1 is displayed as "SP1", and standby 1 is displayed as "WT1".



NOTE
Data registered becomes effective after the screen display is switched to the tool management screen.

10.3.3 Input of Customize Data with the Decimal Point

With the function for input of customize data with the decimal point, the number of decimal places can be set using the G10 format for each customize data item (customize data 1, ..., 40) to enable data input with the decimal point.

Format

G10L77P5;
N_R_;
G11;
N_: Customize data number
R_: Decimal point position

Explanation

- **Customize data number (N_)**

- A value of 1 to 4 can be specified for N.
- When tool management function customize data extension (5 to 20) is available, a value of 1 to 20 can be specified.
- When tool management function customize data extension (5 to 40) is available, a value of 1 to 40 can be specified.

- **Decimal point position (R_)**

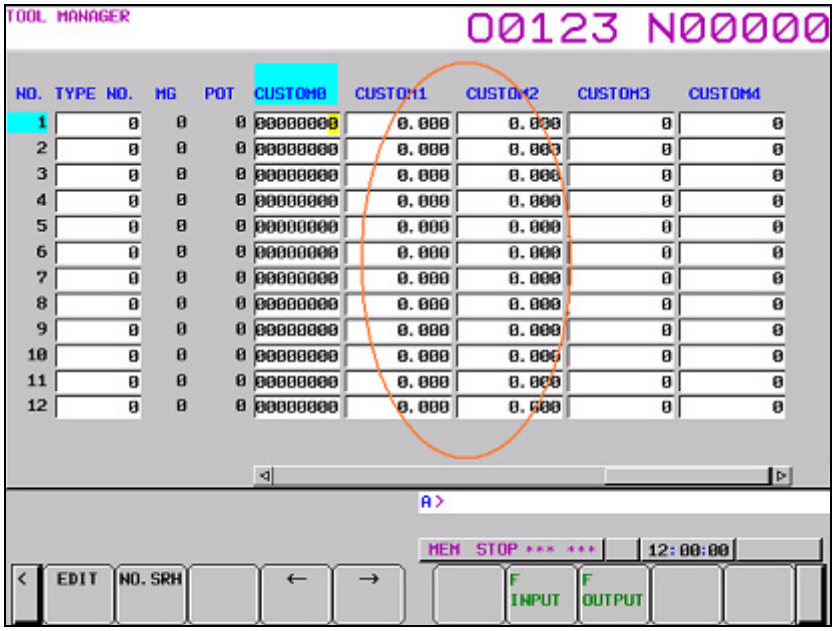
Specify a decimal point position (0 to 7). When 0 is set, the number of decimal places of customize data cannot be input.

NOTE
1 If G10 L77 P5 is terminated normally, the power must be turned off before operation is continued.
2 The setting becomes effective after the power is turned off then back on.
3 When the number of decimal places is set for customize data, the number of decimal places can be input only through the MDI keys.
4 To set the number of decimal places for customize data 5 to 20, the customize data extension (5 to 20) option for the tool management function or the customize data extension (5 to 40) option for the tool management function is required.
5 To set the number of decimal places for customize data 21 to 40, the customize data extension (5 to 40) option for the tool management function is required.

Example 1

When customize data 1 and customize data 2 are input with three decimal places

G10L77P5;	Set the number of decimal places for customize data
N1 R3;	Set the number of decimal places to 3 for customize data 1
N2 R3;	Set the number of decimal places to 3 for customize data 2
G11;	Cancel the setting mode



- Use in the G10 format

Between G10 and G11, the decimal point cannot be specified at any address. So, if decimal point input for customize data is enabled with this function, a G10 L75/L76/L77 command using a custom macro variable (#8431 to #8450) reads and sets the value of the integer part.

Example 2

(Example 1)

Condition:

"3" is set as the decimal point position of customize data 1.

"1" is set as the decimal point position of customize data 2.

Operation:

Data is transferred from customize data 1 to customize data 2 by using a custom macro variable.

```
G10L77P5;          <1> Set customize data decimal point position
N1 R3;             <2> Set 3 as decimal point position of customize data 1
N2 R1;             <3> Set 1 as decimal point position of customize data 2
G11;               <4> Cancel the setting mode
;
G10 L75 P1;        <5> Register tool management data
N01;               <6> Register with No. 1
P1 R12345;         <7> Set “12.345” for customize data 1
P2 R#8431;         <8> Set “1.2” for customize data 2
G11;               <9> Cancel the setting mode
```

In Example 1, customize data 1 is directly set in customize data 2 by using a custom macro variable. Customize data 1 holds "12.345". In step <8>, only the integer part is read and processed as "P2 R12".
So, "1.2" is set in customize data 2.

(Example 2)

Condition:

"3" is set as the decimal point position of customize data 1.
"1" is set as the decimal point position of customize data 2.

Operation:

By using the custom macro variable of customize data 1, the data is transferred to customize data 2 after being multiplied by 1000.

```
G10L77P5;          <1> Set customize data decimal point position
N1 R3;             <2> Set 3 as decimal point position of customize data 1
N2 R1;             <3> Set 1 as decimal point position of customize data 2
G11;               <4> Cancel the setting mode
;
G10 L75 P1;        <5> Register tool management data
N01;               <6> Register with No. 1
P1 R12345;         <7> Set “12.345” for customize data 1
P2 R[#8431*1000];  <8> Set “1234.5” for customize data 2
G11;               <9> Cancel the setting mode
```

In Example 2, customize data 1 is directly set in customize data 2 by using a custom macro variable. Customize data 1 holds "12.345". In step <8>, the data is multiplied by 1000 to eliminate the fractional part.
So, this command is equivalent to "P2 R12345", so that "1234.5" is set in customize data 2.

10.3.4 Protection of Various Tool Information Items with the KEY Signal

When tool management data is in the edit state, various information items can be modified. By setting bit 0 of parameter No. 13204 to 1, tool management data can be protected with the KEY signal so that various information items are not registered, modified, and deleted.

10.3.5 Selection of a Tool Life Count Period

A tool life count period can be chosen between 1 sec and 8 msec on a tool-by-tool basis.

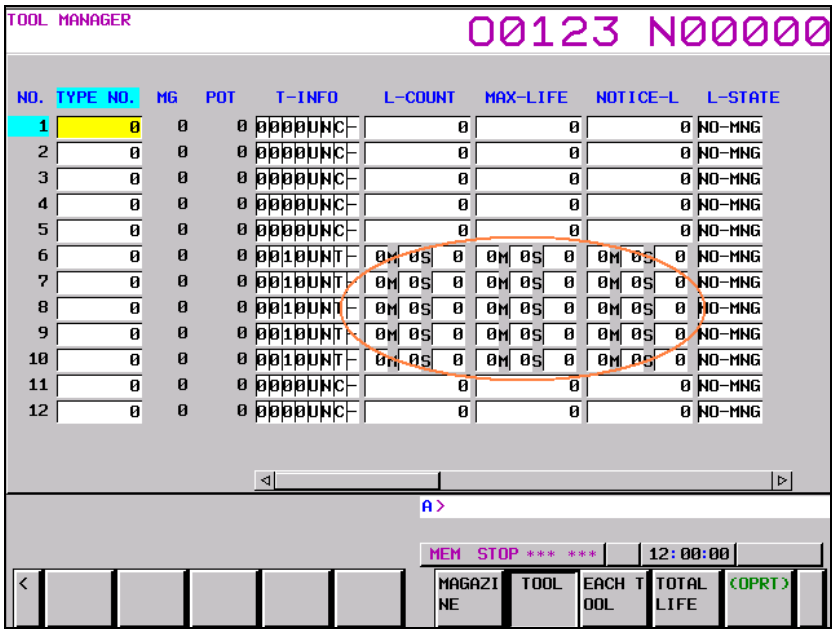
- Tool life count period selection

Bit 5 of tool information is used to make a life count period selection.

Item		Description
Data length		1 byte (flag data)
#5	REV	0: A life count period of 1 sec is used. (S) 1: A life count period of 8 msec is used. (M)

Range of count is as follows.
1sec: 0 to 3,599,999 seconds (999 hours 59 minutes 59 seconds)
8msec: 0 to 3,599,992 ms (59 minutes 59 seconds 992 milliseconds)

NOTE
This function is valid when the tool information TIM (#1) is set to 1.



10.3.6 Each tool Data Screen

All data for a specified tool can be extracted and displayed.

10.3.7 Total Life Time Display for Tools of The Same Type

The remaining lives of tools with the same type numbers are totaled, and totals are displayed in order by tool type number or by remaining life. Also, data of tools with the same tool type numbers are displayed in a list.

10.4 TOOL MANAGEMENT FUNCTION FOR OVERSIZE TOOLS

Overview

Tool management function for oversize tools is added to the tool management function.
The figure of an oversize tool can be defined freely, and the figure of each oversize tool is registered. When an oversize tool is stored in a cartridge, interference with tools stored in other pots is considered. This function is usable with cartridges of chain type and matrix type.

Format

By G10 data input, an oversize tool number can be set in the tool management data. Moreover, tool figure data can be set.

<Registration of new tool management data>

G10 L75 P1 ;
N_;
A_;
G11 ;
N_: Tool management data number
A_: Specify tool figure number (0 to 20).

<Modification of tool management data>

G10 L75 P2;
N_;
A_;
G11 ;
N_: Tool management data number
A_: Specify tool figure number (0 to 20).

<Registration of tool figure data>

G10 L77 P6 ;
N_P_Q_R_S_T_;
G11 ;
N_: Tool figure number
P_: Number of pots to be occupied in left direction (in steps of 0.5 pot)
Q_: Number of pots to be occupied in right direction (in steps of 0.5 pot)
R_: Number of pots to be occupied in upper direction (in steps of 0.5 pot)
S_: Number of pots to be occupied in lower direction (in steps of 0.5 pot)
T_: 0 for figure A, or 1 for figure B

NOTE
1 If a target tool is registered in a cartridge and interferes with other tools in registration or modification of tool figure data of the tool management data, alarm PS5360, “TOOL INTERFERENCE CHECK ERROR” is issued. (The data is not input.)
2 If a tool interferes with other tools in registration or modification to the cartridge management table, alarm PS5360 is issued. (The data is not input.)
3 If an attempt is made to modify tool figure data, and the tool for which the tool figure data number subject to modification is set is registered in the cartridge, alarm PS5360 is issued. (The data is not input.)

10.5 TOOL LIFE MANAGEMENT

Tools are classified into several groups, and a tool life (use count or use duration) is specified for each group in advance. Each time a tool is used, its life is counted, and when the tool life expires, a new tool that is sequenced next within the same group is selected automatically. With this function, the tool life can be managed while machining is being performed continuously. Data for tool life management consists of tool group numbers, tool life values, tool numbers, and codes for specifying a tool offset value. These data items are registered in the CNC.

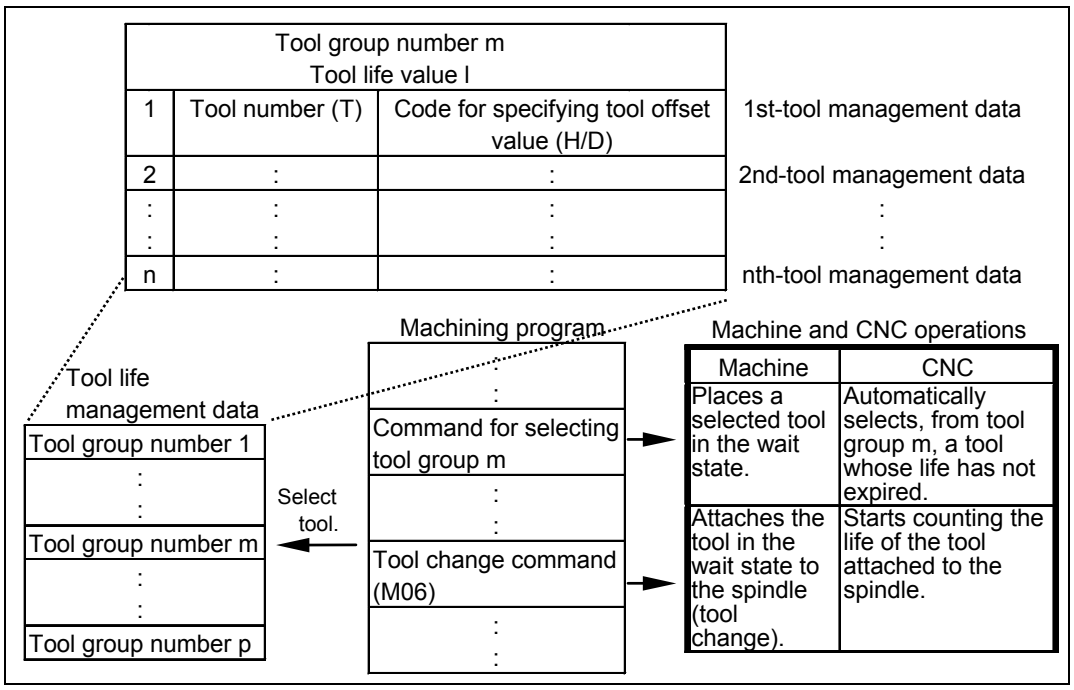


Fig. 10.5 (a) Tool selection from machining program

M

A group is selected by a T code, and tool life counting is started by the M06 command.

T

The T series has two tool change types (the turret type and the ATC type). The turret type uses only a T code to select a group, specify a tool offset value, and start tool life counting, whereas the ATC type, like the M series, uses a T code to select a group and the M06 command to start tool life counting. With the ATC type, only a D code is used for specifying a tool offset value. The tool change type is selected by bit 3 (TCT) of parameter No. 5040.

> ⚠ **CAUTION**
> This function cannot be used if bit 1 (LGN) of parameter No. 5002 is set to 1 to use the same number as the tool selection number to specify a geometric offset number.

- Life management B function

If the tool life management B function is enabled, the maximum tool life value can be extended, and the tool life expiration prior notice signal can be output to post tool life expiration in advance when the remaining life (the life value minus the life counter value) has reached the remaining life setting. The remaining life setting is registered as tool life management data in the CNC in advance.
The tool life management B function is enabled by setting bit 4 (LFB) of parameter No. 6805 to 1.

M

If the tool life management B function is enabled, the function for selecting a tool group by an arbitrary group number can be used.

T

The tool life management B function can be used. However, the function for selecting a tool group by an arbitrary group number can be used only if the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1).

- Maximum number of tool life management groups and multipath system

Up to 256 tool life management groups can be used in the entire CNC system.
For each path, set a maximum number of groups to be used in parameter No. 6813.
The maximum number of groups must be a multiple of the minimum number of groups (eight groups). If the maximum number of groups is 0, the tool life management function is disabled.
The total number of groups in the entire CNC system can be expanded to 1024 groups by using an option for additional tool life management groups.

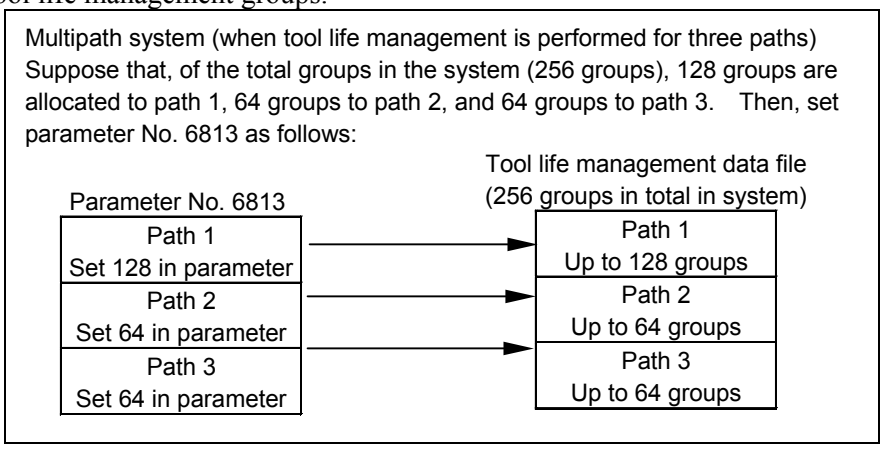


Fig. 10.5 (b) Group allocation in entire system

> **⚠ CAUTION**
> Re-setting this parameter results in the tool life management data file being cleared at power-on. Once the parameter is re-set, register tool life management data for each path again.

10.5.1 Tool Life Management Data

Tool life management data consists of tool group numbers, tool numbers, codes for specifying a tool offset value, tool life values, arbitrary group numbers, and remaining life settings.
Whether to use arbitrary group numbers and remaining life settings can be specified by setting bit 5 (TGN) and bit 3 (GRP) of parameter No. 6802.

Explanation

- Tool group number

Up to 256 tool life management groups can be used in the entire CNC system.
In parameter No. 6813, set a maximum number of groups to be used. With the set maximum number of groups, up to four tools can be registered per group. The combination of the maximum number of registrable groups and the maximum number of tools can be changed by setting bits 0 (GS1) and bit 1 (GS2) of parameter No. 6800 as follows.

Table 10.5.1 Maximum numbers of registrable groups and tools

Bit 1 (GS2) of parameter No. 6800	Bit 0 (GS1) of parameter No. 6800	Number of groups	Number of tools
0	0	1/8 of maximum number of groups (parameter No. 6813)	32
0	1	1/4 of maximum number of groups (parameter No. 6813)	16
1	0	1/2 of maximum number of groups (parameter No. 6813)	8
1	1	Maximum number of groups (parameter No. 6813)	4

⚠ CAUTION
After changing the setting of bits 0 (GS1) and 1 (GS2) of parameter No. 6800, re-register tool life management data by issuing the G10L3 (registration after deletion of data for all groups).
Otherwise, the newly set combination does not become valid.

NOTE
The total number of groups in the entire CNC system can be expanded to up to 1024 groups, using an option for additional tool life management groups.

- Tool number

A tool number is specified with a T code. A number consisting of up to eight digits (99999999) can be specified.

NOTE
The maximum number of digits usable in the T code is specified in parameter No. 3032.

- Codes for specifying a tool offset value

M

Codes for specifying a tool offset value include an H code (for tool length offset) and a D code (for cutter compensation). Numbers up to 999 (up to three digits long) can be registered as codes for specifying tool offset values. However, the number of registered codes must not exceed the number of tool offsets the CNC can use.

NOTE
If codes for specifying tool offset values are not used, the registration of these codes can be omitted.

T

If the tool change type is the turret type (bit 3 (TCT) of parameter No. 5040 = 0), neither the H code nor D code is used to specify a tool offset value. The T code includes a compensation code. If the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1), only the D code is used.

- Tool life value

A tool life value can be registered in terms of use duration or use count. The maximum value is as follows:
Up to 4300 minutes can be registered if duration specification is selected, or up to 65535 times can be registered if use count specification is selected.
When the tool life management B function is enabled, up to 100000 minutes (or 60000 minutes if the life is counted every 0.1 second) can be registered if duration specification is selected, or up to 99999999 times can be registered if use count specification is selected.

- Arbitrary group number

M

If a function for allowing specification of arbitrary group numbers is used (bit 5 (TGN) of parameter No. 6802 = 1), an arbitrary group number can be specified with a T code to select a tool life management group.
An arbitrary group number ranging from 1 to 99999999 can be specified.

T

Arbitrary group numbers can be used only if the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1).

- Remaining life setting

Bit 3 (GRP) of parameter No. 6802 is used to specify which value, a value set for each group or a parameter-set value (parameters Nos. 6844 and 6845), is to be used as the remaining life setting until a new tool is selected.
If a value set for each group is used (bit 3 (GRP) of parameter No. 6802 = 1), the value is registered as tool life management data.
A value ranging from 1 to 100000 minutes (or 60000 minutes if the life is counted every 0.1 second) can be specified if duration specification is selected, or a value up to 99999999 times can be specified if use count specification is selected.

10.5.2 Registering, Changing, and Deleting Tool Life Management Data

By programming, tool life management data can be registered in the CNC, and registered tool life management data can be changed or deleted.

Explanation

The program format varies depending on the following four types of operation:

- Registration after deletion of all groups

After all registered tool life management data is deleted, programmed tool life management data is registered.

- Change of tool life management data

Tool life management data can be set for a group for which no tool life management data is registered, and already registered tool life management data can be changed.

- Deletion of tool life management data

Tool life management data can be deleted.

- Setting of tool life count type

A count type (use duration or use count) can be set for each group separately.

Format

- Registration after deletion of all groups

M

Format	Meaning
G10L3; P-L-; T-H-D-; T-H-D-; : P-L-; T-H-D-; T-H-D-; : G11; M02(M30);	G10L3: Register data after deleting data of all groups. P-: Group number L-: Tool life value T-: Tool number H-: Code for specifying tool offset value (H code) D-: Code for specifying tool offset value (D code) G11: End of registration

T

Format	Meaning
G10L3; P-L-; T-(D-); T-(D-); : P-L-; T-(D-); T-(D-); : G11; M02(M30);	G10L3: Register data after deleting data of all groups. P-: Group number L-: Tool life value T-: For turret type (bit 3 (TCT) of parameter No. 5040 = 0), tool number and tool offset number For ATC type (bit 3 (TCT) of parameter No. 5040 = 1), tool number G11: End of registration (D-): For ATC type (bit 3 (TCT) of parameter No. 5040 = 1), code for specifying tool offset value

NOTE
No H code is used even if the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1).

If more than one offset value is to be used for the same tool within one process, specify the command as shown below (on the assumption that tool offset numbers are two digits long).

Format	Meaning
G10L3; P-L-; T0101; T0102; T0103; : G11; M02(M30);	 Tool number 01, tool offset number 01 Tool number 01, tool offset number 02 Tool number 01, tool offset number 03

- Change of tool life management data

M

Format	Meaning
G10L3P1; P-L-; T-H-D-; T-H-D-; : P-L-; T-H-D-; T-H-D-; : G11; M02(M30);	G10L3P1: Start changing group data. P-: Group number L-: Tool life value T-: Tool number H-: Code for specifying tool offset value (H code) D-: Code for specifying tool offset value (D code) G11: End of change of group

T

Format	Meaning
G10L3P1; P-L-; T-(D-); T-(D-); : P-L-; T-(D-); T-(D-); : G11; M02(M30);	G10L3P1: Start changing group data. P-: Group number L-: Tool life value T-: For turret type (bit 3 (TCT) of parameter No. 5040 = 0), tool number and tool offset number For ATC type (bit 3 (TCT) of parameter No. 5040 = 1), tool number G11: End of registration (D-): For ATC type (bit 3 (TCT) of parameter No. 5040 = 1), code for specifying tool offset value

NOTE
No H code is used even if the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1).

- Deletion of tool life management data

Format	Meaning
G10L3P2; P-; P-; P-; P-; : G11; M02(M30);	G10L3P2: Start deleting group data. P-: Group number G11: End of deletion

- Setting of tool life count type

Format	Meaning
G10L3 (or G10L3P1); P-L-Q-; T-H-D-; T-H-D-; : G11; M02(M30);	Q: Life count type (1: Use count. 2: Duration.)

> ⚠ **CAUTION**
> If the Q command is omitted, the life count type is set according to the setting of bit 2 (LTM) of parameter No. 6800.

- Arbitrary group number

M

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1), and use of the function for specifying an arbitrary group number is set (bit 5 (TGN) of parameter No. 6802 = 1), an arbitrary group number can be registered by programming a command.

Format	Meaning
G10L3; (or G10L3P1); P-L-T-; T-H-D-; T-H-D-; : G11; M02(M30);	T-: Arbitrary group number (1 to 99999999)

> **NOTE**
> 1 A group number that is the same as the group number of another group cannot be registered.
> An attempt to register such a group number results in alarm PS0431, “ILLEGAL T/R DATA OF TOOL LIFE”.
> 2 If an arbitrary group number (T) is 0, alarm PS0431 is issued.
> 3 If ever an arbitrary group number is specified, the T command must be specified. If the T command is omitted, alarm PS0149, “FORMAT ERROR IN G10L3” is issued.

T

The function for specifying an arbitrary group number is available only if the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1).
If the tool change type is the ATC type, the format used for setting an arbitrary group number is the same as for the M series.

Remaining life setting

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1), and a remaining life value is to be set for each group separately (bit 3 (GRP) of parameter No. 6802 = 1), a remaining life value can be set as tool life management data by programming a command.

Format	Meaning
G10L3; (or G10L3P1); P-L-R-; T-H-D-; T-H-D-; : G11; M02(M30);	R-: Remaining life setting (remaining life until a new tool is selected)

NOTE
1 If the remaining life setting (R) is 0 or omitted, the remaining life is assumed to be 0. In this case, the tool life expiration prior notice function is disabled.
2 The remaining life setting (R) cannot exceed the life value (L). If an attempt is made to specify a value greater than the life value, alarm PS0431 is issued.
3 The remaining life setting (R) must be an unsigned value having no decimal point.
4 The unit of the remaining life setting (R) depends on the setting of bit 1 (FGL) of parameter No. 6805, which selects the unit of the tool life value.

Tool life value

- If the life management B function is disabled

If the tool life management B function is disabled (bit 4 (LFB) of parameter No. 6805 = 0), a tool life value is registered as a duration or a use count according to the setting of bit 2 (LTM) of parameter No. 6800 or the setting of a count type (Q command). The maximum values are listed below.

Table 10.5.2 (a) Life count types and maximum life values

Bit 4 (LFB) of parameter No. 6805	Bit 2 (LTM) of parameter No. 6800	Life count type	Maximum life value
0	0	Use count specification	65535 times
	1	Duration specification	4300 minutes

If the count type is duration specification, the unit of a life value specified with address L in a program may be one minute or 0.1 second, which is determined by the setting of bit 1 (FGL) of parameter No. 6805.

Table 10.5.2 (b) Life value unit and maximum value in L command

Bit 4 (LFB) of parameter No. 6805	Bit 1 (FGL) of parameter No. 6805	Life value unit	Maximum value in L command	Example
0	0	1 minute	4300	L100: Life value is 100 minutes.
	1	0.1 second	2580000	L1000: Life value is 100 seconds.

- If the life management B function is enabled

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1)

Table 10.5.2 (c) Life count types and maximum life values

Bit 4 (LFB) of parameter No. 6805	Bit 2 (LTM) of parameter No. 6800	Life count type	Maximum life value
1	0	Use count specification	99999999 times
	1	Duration specification	100000 minutes
			60000 minutes(Note)

Table 10.5.2 (d) Life value unit and maximum value in L command

Bit 4 (LFB) of parameter No. 6805	Bit 1 (FGL) of parameter No. 6805	Life value unit	Maximum value in L command	Example
1	0	1 minute	100000	L100: Life value is 100 minutes.
			60000(Note)	
	1	0.1 second	60000000	L1000: Life value is 100 seconds.
			36000000(Note)	

NOTE
If the life is counted every 0.1 second (bit 0 (FCO) of parameter No. 6805 = 1), the maximum value specified by duration is 60000 minutes.

10.5.3 Tool Life Management Commands in Machining Program

Explanation

M

- Commands

The following commands are used for tool life management:

TOOOOOOOO;

Specifies a tool group number.
The tool life management function selects, from a specified group, a tool whose life has not expired, and outputs its T code signal.
In OOOOOOOO, specify the sum of the tool life management ignore number specified in parameter No. 6810 and a desired group number.
Example:

To specify tool group number 1 when the tool life management ignore number is 100, specify "T101;".

NOTE
If OOOOOOOO is not greater than a tool life management ignore number, the T code is treated as an ordinary T code.

If an arbitrary group number is used (bit 4 (LFB) of parameter No. 6805 = 1, and bit 5 (TGN) of parameter No. 6802 = 1), it must be a preset arbitrary group number rather than a tool group number specified in OOOOOOOO.
Example:

To specify tool group number 1 when arbitrary group number 1234 is set for tool group number 1, specify T1234.

NOTE
1 If an arbitrary group number is used, the tool life management ignore number specified in parameter No. 6810 is not used.
2 If the group specified by OOOOOOOO is not present, the CNC assumes that a tool not controlled under the tool life management function is selected.

M06;

Terminates life management for the previously used tools, and begins counting the life of a new tool selected with the T code.

NOTE
1 M06 is treated as an M code not involving buffering.
2 If multiple M codes are to be specified in the same block, M06 can be specified anywhere in the M codes.

H99;

Selects the H code registered in tool life management data for the currently used tool to enable tool length offset. Parameter No. 13265 can be used to enable compensation according to an H code other than H99.

H00;

Cancels tool length offset.

D99;

Selects the D code registered in tool life management data for the currently used tool to perform cutter compensation.
Parameter No. 13266 can be used to enable compensation according to a D code other than D99.

D00;

Cancels cutter compensation.

NOTE
H99 and D99 must be specified after the M06 command. If a code other than H99 or D99 or a code other than the H or D code set, respectively, in parameter No. 13265 or 13266 is specified following M06, no H code or D code is selected for tool life management data.

- Types

For tool life management, the four tool change types (types A to D) listed below are used. Which type to use varies from one machine to another. For details, refer to the relevant manual of each machine tool builder.

Table 10.5.3 Differences among tool change types

Tool change type	A		B		C		D	
Parameters M6T and M6E	M6T	M6E	M6T	M6E	M6T	M6E	M6T	M6E
M6T (No.6800#7) M6E (No.6801#7)	0	0	1	0	1	0		1
Tool group number specified in the same block as the tool change command (M06)	Tool group already used		Tool group to be used next					
Time when tool life is counted	Life counting is performed for a tool in the specified tool group if M06 is specified next.						Life counting is performed if a tool in the tool group specified in the same block as M06 is specified.	
Remarks	If the T command (retract tool group) following the M06 command is not the currently used tool group, alarm PS0155, “ILLEGAL T-CODE COMMAND” is issued (if bit 6 (IGI) of parameter No. 6800 = 0).		Normally, if a tool group number command is specified alone, type B is used. However, even if a tool group number command is specified alone with type C, no alarm is issued. (This means that there is no difference in operation between types B and C.)				If M06 is specified alone, alarm PS0153, “T-CODE NOT FOUND” is raised.	

NOTE
If a tool group number is specified and a new tool is selected, the new tool selection signal is output.

- Tool length compensation in tool axis direction
- Tool center point control

For these functions, a compensation value selected in tool life management is used for compensation.

T

If the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1), commands are specified in the same manner as for the M series except that neither H99 nor H00 is used for the T series. See the description for the M series. For details on the tool compensation types applied when the tool change type is the ATC type, refer to Section, "EXTENDED TOOL SELECTION" of the OPERATOR'S MANUAL (Lathe System) (B-64484EN-1).
The command specification method for the turret type (bit 3 (TCT) of parameter No. 5040 = 0) is explained below.

> ⚠ **CAUTION**
> This function cannot be used if the same number as the tool selection number is to be used to specify geometry offset number (bit 1 (LGN) of parameter No. 5002 = 1).

- Commands

TOOOOOO99;

The tool life management function ends counting the life of the tool used so far, selects, from the group specified by OOOOOO, a tool whose life has not expired, outputs the T code signal for the tool, and starts counting the life of the tool.

Example:

Suppose that a T199 command (with a tool offset specified with the lower two digits) is issued to cause the life management function to select T10001 of tool group 1. Then, T code 100 is output, and the tool offset number 1 is selected.

If the condition that a new tool be selected is not met, and the second or subsequent selection of the same group is made since the entry of the control unit into the automatic operation start state from the reset state, the next to the currently selected T code is selected if more than one offset is registered.
If the third selection is made, for example, the third offset is selected from among multiple offsets registered for the same tool.

Example:

Suppose that the tool offset is specified with the lower two digits, and that two offset numbers are set for the same tool number in group 1 with the following two T codes:

T10001 T10002

The first T199 command issued since the entry of the control unit into the automatic operation start state from the reset state selects the first T code, T10001. Then, if T199 is issued again before the control unit is reset, the second T code, T10002, is selected. Furthermore, if T199 is issued again before the control unit is reset, the second T code, T10002, is selected because no third offset is present.
Setting bit 1 (TSM) of parameter No. 6801 set to 1 enables life counting to be performed for each T code separately even if T codes specifying multiple offsets for the same number are registered.

TOOOOOO88;

The offset of the tool whose life is currently counted by tool life management is canceled. The tool offset code is set to 00, and the tool number is output as a T code signal.

Example:

Suppose that the tool number of the tool currently used by the life management function is 100. Then, issuing a T188 command (with a tool offset specified with the lower two digits) outputs T code 100 and selects offset number 0, canceling the offset.

NOTE
If life counting is not performed, or if the specified tool does not belong to the group for which life counting is being performed, alarm PS0155 is issued.

The numbers of digits in OOOOOO and 99/88 vary as follows:

No.5028	99	88
1	TOOOOOOO 9 ↑ ↑ Select group Start life counting	TOOOOOOO 8 ↑ ↑ Select group Cancel tool offset
2	TOOOOOO 99 ↑ ↑ Select group Start life counting	TOOOOOO 88 ↑ ↑ Select group Cancel tool offset
3	TOOOOO 999 ↑ ↑ Select group Start life counting	TOOOOO 888 ↑ ↑ Select group Cancel tool offset

The maximum number of digits in T codes is set in parameter No. 3032.
The number of digits used for specifying an offset number is selected by parameter No. 5028. If 0 is selected, the number of digits depends on the number of tool offsets.
Example:

If there are one to nine tool offsets:	Lowest digit
If there are 10 to 99 tool offsets:	Lower two digits
If there are 100 to 999 tool offsets:	Lower three digits

NOTE
Offset start and cancel operations involve compensation by moving a tool or by shifting the coordinate system. Using bit 6 (LWM) of parameter No. 5002 can select whether to perform a compensation operation if a T code is specified or if an axis move command is specified.
For details, refer to refer to Subsection, “Offset” of the OPERATOR’S MANUAL (Lathe System) (B-64484EN-1).

TOOOOOOΔΔ;

If the tool offset number in ΔΔ is neither 99 nor 88, the T code is treated as an ordinary T code. If life counting is being currently performed, it is ended.

Examples

M

- Tool change type A

If a block specifying a tool change command (M06) also contains a tool group command (T code), the T code is used as a command for returning the tool to its cartridge. By specifying a tool group number with a T code, the number of the tool used so far is output as a T code signal. If the specified tool number is not a tool number of the tool group of the tool being used, alarm PS0155 is issued. The alarm, however, can be suppressed by setting bit 6 (IGI) of parameter No. 6800 to 1.

Example: Suppose that the tool life management ignore number is 100.	
T101 ;	A tool whose life has not expired is selected from group 1.
:	(Suppose that tool number 010 is selected.)
M06 ;	Tool life counting is performed for the tool in group 1.
:	(The life of tool number 010 is counted.)
T102 ;	A tool whose life has not expired is selected from group 2.
:	(Suppose that tool number 100 is selected.)
M06 T101 ;	Tool life counting is performed for the tool in group 2.
:	(The life of tool number 100 is counted.)
:	The number of the tool currently used (in group 1) is output with a T code signal. (Tool
:	number 010 is output.)
T103 ;	A tool whose life has not expired is selected from group 3.
:	(Suppose that tool number 200 is selected.)
M06 T102 ;	Tool life counting is performed for the tool in group 3.
:	(The life of tool number 200 is counted.)
G43 H99 ;	Tool length offset for the tool selected from group 3 is used.
:	
G41 D99 ;	Cutter compensation for the tool selected from group 3 is used.
:	
D00 ;	Cutter compensation is canceled.
:	
H00 ;	Tool length offset is canceled.

- Tool change types B and C

If a block specifying a tool change command (M06) also contains a tool group command (T code), the T code is used to specify a tool group number for which life counting is to be performed by the next tool change command.

Example: Suppose that the tool life management ignore number is 100.	
T101 ;	A tool whose life has not expired is selected from group 1.
:	(Suppose that tool number 010 is selected.)
M06 T102 ;	Tool life counting is performed for the tool in group 1.
:	(The life of tool number 010 is counted.)
:	A tool whose life has not expired is selected from group 2.
:	(Suppose that tool number 100 is selected.)
M06 T103 ;	Tool life counting is performed for the tool in group 2.
:	(The life of tool number 100 is counted.)
:	A tool whose life has not expired is selected from group 3.
:	(Suppose that tool number 200 is selected.)
G43 H99 ;	Tool length offset for the tool selected from group 2 is used.
:	
G41 D99 ;	Cutter compensation for the tool selected from group 2 is used.
:	
D00 ;	Cutter compensation is canceled.
:	
H00 ;	Tool length offset is canceled.
:	
M06 T104 ;	Tool life counting is performed for the tool in group 3.
:	(The life of tool number 200 is counted.)
	A tool whose life has not expired is selected from group 4.

- Tool change type D

For a tool selected by a tool group command (T code), life counting is performed by a tool change command (M06) specified in the same block as the tool group command. Specifying a T code alone does not results in an alarm; however, specifying an M06 command alone results in alarm PS0153. However, the alarm can be suppressed by setting bit 7 (TAD) of parameter No. 6805 to 1.

Example: Suppose that the tool life management ignore number is 100.	
T101 M06 ;	A tool whose life has not expired is selected from group 1.
:	(Suppose that tool number 010 is selected.)
:	Tool life counting is performed for the tool in group 1.
:	(The life of tool number 010 is counted.)
T102 M06 ;	A tool whose life has not expired is selected from group 2.
:	(Suppose that tool number 100 is selected.)
:	Tool life counting is performed for the tool in group 2.
:	(The life of tool number 100 is counted.)
G43 H99 ;	Tool length offset for the tool selected from group 2 is used.
:	
G41 D99 ;	Cutter compensation for the tool selected from group 2 is used.
:	
D00 ;	Cutter compensation is canceled.
:	
H00 ;	Tool length offset is canceled.
:	
T103 M06 ;	A tool whose life has not expired is selected from group 3.
:	Tool life counting is performed for the tool in group 3.

T

For the ATC type (bit 3 (TCT) of parameter No. 5040 = 1), commands are specified in the same manner as for the M series except that D99 rather than H99 is used. For the ATC type, see the description for the M series.

The following shows how commands are specified if the turret type is used (bit 3 (TCT) of parameter No. 5040 = 0):

Example: Suppose that offset numbers are two digits long.	
T0199 ;	A tool whose life has not expired is selected from group 1.
:	(Suppose that T1001 is selected. The tool number is 10, and the offset number is 01.)
:	
:	Tool life counting is performed for the tool in group 1.
:	(The life of tool number 10 is counted.)
:	
T0188 ;	The offset of the tool being used in group 1 is canceled.
:	(Since the tool being used is T1001, the tool number is 10, and the offset number is 00.)
:	
T0299 ;	A tool whose life has not expired is selected from group 2.
:	(Suppose that T2002 is selected. The tool number is 20, and the offset number is 02.)
:	Tool life counting is performed for the tool in group 2.
:	(The life of tool number 20 is counted.)
:	
T0299 ;	If more than one offset number is specified for the currently used tool in group 2, the next
:	offset number is selected.
:	(Suppose that T2002 and T2003 are registered with tool number 20. In this case, T2003 is
:	selected. The tool number is 20, and the offset number is 03.)
:	
T0301 ;	Life counting for the tool in group 2 ends, and this command is treated as an ordinary T
:	code.
:	(The tool number is 03, and the offset number is 01.)

10.5.4 Tool Life Counting and Tool Selection

Either use count specification or duration specification is selected as the tool life count type according to the state of bit 2 (LTM) of parameter No. 6800. Life counting is performed for each group separately, and the life counter contents are preserved even after the power is turned off.

Table 10.5.4 Tool life management count types and intervals

Tool life count type	Use count specification	Time specification
Bit 2 (LTM) of parameter No. 6800	0	1
Life count interval	Incremented by one for tools used in one program Counting can be resumed by using tool life count restart M code (parameter No. 6811).	Bit 0 (FCO) of parameter No. 6805 0: Every second 1: Every 0.1 second Can be changed by override.

Explanation

M

- Use count specification (LTM=0)

If a tool group (T code) is specified, a tool whose life has not expired is selected from the specified tool group. The life counter for the selected tool is then incremented by one by a tool change command (M06). Unless a tool life count restart M code is specified, selection of a new tool and the increment operation can be performed only if a tool group number command and a tool change command are issued for the first time since the entry of the control unit into the automatic operation start state from the reset state.

> ⚠ **CAUTION**
> No matter how many times the same tool group number is specified in a program, the use count is not incremented, and no new tool is selected.

- Duration specification (LTM=1)

After all registered tool life management data is deleted, programmed tool life management data is registered.
If a tool group command (T code) is specified, a tool whose life has not expired is selected from the specified tool group. Then, life management for the selected tool is started by a tool change command (M06). Life management (counting) is performed by measuring the time during which the tool is actually used in the cutting mode at regular intervals (every second or 0.1 second). The life count interval is specified by bit 0 (FCO) of parameter No. 6805. The time required for single block stop, feed hold, rapid traverse, dwell, machine lock, and interlock operations is not counted.
Setting bit 2 (LFV) of parameter No. 6801 enables the life count to be overridden as directed by tool life count override signals. An override from 0 times to 99.9 times can be applied. If 0 times is specified, counting is not performed.

T

If the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1), the same specification method as for the M series applies. See the description for the M series.
The following explains the specification method for the turret type (bit 3 (TCT) of parameter No. 5040 = 0):

- Use count specification (LTM=0)

If a tool group command (T○○99 code) is issued, a tool whose life has not expired is selected from the specified tool group, and the life counter for the selected tool is incremented by one. Unless a tool life count restart M code is specified, selection of a new tool and the increment operation can be performed only if a tool group number command and a tool change command are issued for the first time since the entry of the control unit into the automatic operation start state from the reset state.

> ⚠ **CAUTION**
> No matter how many times the same tool group number is specified in a program, the use count is not incremented, and no new tool is selected.

- Duration specification (LTM=1)

If a tool group command (T○○99 code) is specified, a tool whose life has not expired is selected from the specified tool group, and tool management for the selected tool starts.
Life management (counting) is performed by measuring the time during which the tool is actually used in the cutting mode at regular intervals (every second or 0.1 second). The life count interval is specified by bit 0 (FCO) of parameter No. 6805. The time required for single block stop, feed hold, rapid traverse, dwell, machine lock, and interlock operations is not counted.
Setting bit 2 (LFV) of parameter No. 6801 enables the life count to be overridden according tool life count override signals. An override from 0 times to 99.9 times can be applied. If 0 times is specified, counting is not performed.

M

T

NOTE

1 When a tool is selected, tools are searched starting from the current tool toward the last tool to find a tool whose life has not expired. When the last tool is reached during this search, the search restarts from the first tool. If the search does find any tool whose life has not expired, the last tool is selected.
When the tool currently used is changed by the tool skip signal, the next new tool is selected using the method described here.
2 If tool life counting indicates that the life of the last tool in a group has expired, the tool change signal is output. If the life count type is duration specification, the signal is output as soon as the life of the last tool in the group has expired. If the life count type is use count specification, the signal is output when the CNC is reset by a command such as M02 or M30 or when the tool life count restart M code is specified after the life of the last tool in the group has expired.
3 If a T command is specified, a group and a tool in the group are selected while the T command is buffered. This means that if a block to be buffered contains a T command specifying a group while machining is being performed with that group selected, the next T command is already buffered even if the tool life expires during machining, so the next tool is not selected. To prevent this, if the life count type is duration specification and a T command is to be specified to select the same group successively, insert an M code for suppressing buffering into a place immediately before the T command.
4 When the tool life is counted, the remaining life of a group (the life value minus the life counter value) is compared with the remaining life setting, and the status of the tool life expiration prior notice signal is changed according to the result of the comparison.

- M99

If the life count is specified by use count and bit 0 (T99) of parameter No. 6802 is 1, the tool change signal TLCH is output and the automatic operation is stopped if the life of at least one tool group has expired when the M99 command is executed. If the life count type is duration specification, the tool change signal is output immediately when the life of at least one tool group has expired; when the M99 command is specified, the automatic operation is stopped, but no more tool change signal is output.

M

If the life count is specified by use count, a tool group command (T code) issued after the M99 command selects, from a specified group, a tool whose life has not expired, and the next tool change command (M06) increments the tool life counter by one.

T

If the life count is specified by use count, when a tool group command (T code) is specified after the M99 command is specified, a tool whose life has not expired is selected from a specified group, and the tool life counter is incremented by one.
If the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1), the same specifications as for the M series apply.

10.5.5 Tool Life Count Restart M Code

Explanation

M

If the life count is specified by use count, the tool change signal is output if the life of at least one tool group has expired when a tool life count restart M code is issued. A tool group command (T code) issued after the tool life count restart M code selects, from a specified group, a tool whose life has not expired, and the next tool change command (M06) increments the tool life counter by one. This enables the tool life to be counted by a tool change command (M06) even if the command is not the first tool change command (M06) issued since the entry of the CNC into the automatic operation start state from the reset state. The tool life count restart M code is specified in parameter No. 6811.

Example:

Suppose that M16 is a tool life count restart M code and that the tool life management ignore number is 100.
Also suppose that the life count is specified by use count.

```
T101 ;  A tool whose life has not expired is selected from group 1.
  :
  :
M06 ;   Tool life management is performed for group 1.
  :       (The tool life counter is incremented by one.)
  :
T102 ;  A tool whose life has not expired is selected from group 2.
  :
  :
M06 ;   Tool life management is performed for group 2.
  :       (The tool life counter is incremented by one.)
  :
M16 ;   Tool life counting is restarted.
T101 ;  A tool whose life has not expired is selected from group 1.
  :
  :
M06 ;   Tool life management is performed for group 1.
  :       (The tool life counter is incremented by one.)
```

T

If the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1), the same specifications as for the M series apply. See the description for the M series.
The following explanation assumes that the tool change type is the turret type (bit 3 (TCT) of parameter No. 5040 = 0).
If the life count is specified by use count, the tool change signal is output if the life of at least one tool group has expired when a tool life count restart M code is specified. A tool group command (T code) issued after the tool life count restart M code selects a tool whose life has not expired from a specified group, and the tool life counter is incremented by one. This enables the tool life to be counted by a tool group command (T code) even if the command is not the first tool group command issued since the entry of the CNC into the automatic operation start state from the reset state. The tool life count restart M code is specified in parameter No. 6811.

Example:

Suppose that M16 is a tool life count restart M code.
Also suppose that the life count is specified by use count.

```
T199 ;  A tool whose life has not expired is selected from group 1.
  :       Tool life management is performed for group 1.
  :       (The tool life counter is incremented by one.)
  :
T299 ;  A tool whose life has not expired is selected from group 2.
  :       Tool life management is performed for group 2.
  :       (The tool life counter is incremented by one.)
  :
M16 ;   Tool life counting is restarted.
T199 ;  A tool whose life has not expired is selected from group 1.
  :       Tool life management is performed for group 1.
  :       (The tool life counter is incremented by one.)
```

M

T

NOTE
1 The tool life count restart M code is treated as an M code not involved in buffering.
2 If the life count type is use count specification, the tool change signal is output if the life of at least one tool group has expired when the tool life count restart M code is specified. If the life count type is duration specification, specifying the tool life count restart M code causes nothing.
3 If multiple M codes are to be specified in the same block, the tool life count restart M code can be specified anywhere in the M codes.

10.5.6 Disabling Life Count

Explanation

M

T

If bit 6 (LFI) of parameter No. 6804 is “1”, the tool life count disable signal LFCIV can be used to select whether to cancel the tool life count.

If the tool life count disable signal LFCIV is “1”, the tool life count disabled signal LFCIF becomes “1”, and the tool life count is disabled.
If the tool life count disable signal LFCIV is “0”, the tool life count disabled signal LFCIF becomes “0”, and the tool life count is enabled.

NOTE
No buffering should occur when the state of the tool life count disable signal LFCIV is changed. So, use, for example, M codes not involved in buffering to change the signal state. If M06 (for the M series) or a tool change T code (for the T series) is issued in a block that follows directly a block in which an auxiliary function code with buffering enabled is used to turn on or off the tool life count disable signal LFCIV, it is likely that a command for specifying whether to count may become incorrect.

10.5.7 Remaining Tool Number Check Function

Overview

M

In tool life management, this function outputs the remaining tool number notice signal if the number of remaining tools in the group selected with a T code command is equal to or less than the value set in parameter No. 6846.

Explanation

M

If, in tool life management, the remaining tool number in the group selected with a T code command is equal to or less than the value set in parameter No. 6846, the remaining tool number notice signal TLAL becomes "1".

If the setting of parameter No. 6846 is 0, the remaining tool number notice signal TLAL is not output.

The remaining tool number notice signal TLAL becomes "0" when:

- A value is input to parameter No. 6846.
- Using a G10 command, registration is performed that involves the erasure of all life management data groups.
- Using a T code command, a group is selected in which the remaining tool number is greater than the setting of parameter No. 6846.

The remaining tool number notice signal TLAL becomes "0" when any of the following is executed in the group that has caused the remaining tool number notice signal TLAL to be generated.

- Clearing of execution data, using the tool change reset signal.
- Change and deletion of tool life management data, using a G10 command.
- Batch deletion of tool groups from the group edit screen of tool life management, addition of tool data to the screen, and deletion from the screen.
- Clearing of tool execution data from the group edit screen of tool life management.
- Clearing of execution data from the list screen of tool life management.
- Execution of any of the FOCAS2 functions given in Table 10.5.7.

Table 10.5.7 FOCAS2 functions that cause TLAL to become "0"

Function name	Description
cnc_clrcntinfo	Clearing of tool life management data (life counter and tool information) (range specification)
cnc_deltlifegrp	Deletion of tool life management data (tool group)
cnc_deltlifedt	Deletion of tool life management data (tool data) (Effective if the group contains only one tool.)
cnc_instlifedt	Addition of tool life management data (tool data)
cnc_wr1tlifedata	Writing of tool life management data (tool data) (Effective if tool information is changed to "expired" or if tool information is changed to "registered" for a tool that is "skipped".)
cnc_wr1tlifedat2	
cnc_wr2tlifedata	

11 AUXILIARY FUNCTION

Overview

There are two types of auxiliary functions; auxiliary function (M code) for specifying spindle start, spindle stop, program end, and so on, and secondary auxiliary function (B code) for specifying index table positioning.
When a move command and auxiliary function are specified in the same block, the commands are executed in one of the following two ways:
(1) Simultaneous execution of the move command and auxiliary function commands.
(2) Executing auxiliary function commands upon completion of move command execution.
The selection of either sequence depends on the machine tool builder's specification. Refer to the manual issued by the machine tool builder for details.

Chapter 11, "AUXILIARY FUNCTION", consists of the following sections:

11.1 AUXILIARY FUNCTION (M FUNCTION) 248
11.2 MULTIPLE M COMMANDS IN A SINGLE BLOCK 249
11.3 M CODE GROUPING FUNCTION.....250
11.4 SECOND AUXILIARY FUNCTIONS (B CODES) 252

11.1 AUXILIARY FUNCTION (M FUNCTION)

When a numeral is specified following address M, code signal and a strobe signal are sent to the machine. The machine uses these signals to turn on or off its functions.
Usually, only one M code can be specified in one block. Depending on the setting of bit 7 (M3B) of parameter No. 3404, up to three M codes can be specified.
Which M code corresponds to which machine function is determined by the machine tool builder. The machine processes all operations specified by M codes except those specified by M98, M99, M198 or called subprogram (parameter No.6071 to 6079), or called custom macro (parameter No.6080 to 6089). Refer to the machine tool builder's instruction manual for details.

Explanation

The following M codes have special meanings.

- M02, M30 (End of program)

This indicates the end of the main program.
Automatic operation is stopped and the CNC unit is reset. (This differs with the machine tool builder.)
After a block specifying the end of the program is executed, control returns to the start of the program.
Bits 5 (M02) and 4 (M30) of parameter No. 3404 can be used to disable M02, M30 from returning control to the start of the program.

- M00 (Program stop)

Automatic operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged. The automatic operation can be restarted by actuating the cycle operation. (This differs with the machine tool builder.)

- M01 (Optional stop)

Similarly to M00, automatic operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator's panel has been pressed.

- M98 (Calling of subprogram)

This code is used to call a subprogram. The code and strobe signals are not sent. See the Section, "SUBPROGRAM (M98, M99)" for details.

- M99 (End of subprogram)

This code indicates the end of a subprogram.
M99 execution returns control to the main program. The code and strobe signals are not sent. See the Section, "SUBPROGRAM (M98, M99)" for details.

11.2 MULTIPLE M COMMANDS IN A SINGLE BLOCK

Usually, only one M code can be specified in one block. By setting bit 7 (M3B) of parameter No. 3404 to 1, however, up to three M codes can be specified simultaneously in one block.
Up to three M codes specified in one block are output to the machine at the same time. So, when compared with a case where a single M code is specified in one block, a reduced machining cycle time can be achieved.

Explanation

CNC allows up to three M codes to be specified in one block. However, some M codes cannot be specified at the same time due to mechanical operation restrictions. For detailed information about the mechanical operation restrictions on simultaneous specification of multiple M codes in one block, refer to the manual of each machine tool builder.
M00, M01, M02, M30, M98, M99, or M198 must not be specified together with another M code.
Some M codes other than M00, M01, M02, M30, M98, M99, and M198 cannot be specified together with other M codes; each of those M codes must be specified in a single block.
Such M codes include these which direct the CNC to perform internal operations in addition to sending the M codes themselves to the machine. To be specified, such M codes are M codes for calling program numbers 9001 to 9009 and M codes for disabling advance reading (buffering) of subsequent blocks. Meanwhile, multiple of M codes that direct the CNC only to send the M codes themselves (without performing internal operations) can be specified in a single block.
However, it is possible to specify multiple M codes that are sent to the machine in the same block unless they direct the CNC to perform internal operations. (Since the processing method depends on the machine, refer to the manual of the machine tool builder.)

Example

One M command in a single block	Multiple M commands in a single block
M40; M50; M60; G28G91X0Y0Z0; : : :	M40M50M60; G28G91X0Y0Z0; : : : : :

11.3 M CODE GROUPING FUNCTION

Overview

Classifying a maximum of 500 M codes into a maximum of 127 groups allows the user:

- To receive an alarm if an M code that must be specified alone is included when multiple M codes are specified in a block.
- To receive an alarm if M codes belonging to the same group are specified in the same block when multiple M codes are specified in a block.

11.3.1 Setting an M Code Group Number Using the Setting Screen

- Procedure for displaying the M code group setting screen

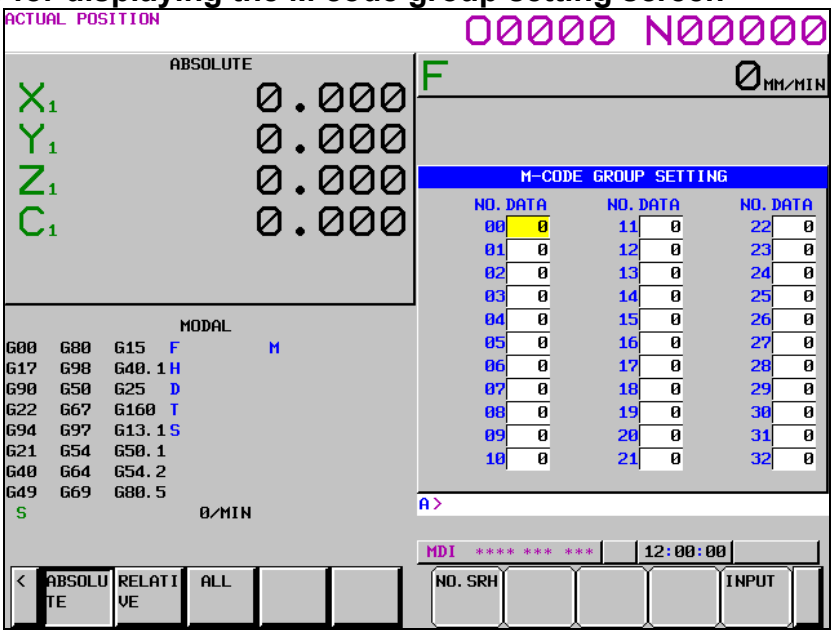


Fig. 11.3 (a) M code group setting screen

You can use the “M code group setting screen (Fig. 11.3 (a))” to set a group number for each M code.

Display the “M code group setting screen” using the following procedure:

(1) Press function key 

and the continuous menu key several times. Soft key [M CODE GROUP] appears.
(2) Press soft key [M CODE GROUP].

In the “NO.” field, M codes for which an M code group can be set are displayed.
An M code group can be set for the following M codes: M00 to M99, and any 400 M codes selected from M100 and subsequent M codes. For details of how to add the 100th and subsequent M codes, see the explanation of parameters Nos. 3441 to 3444.

In the “DATA” field, the M code group number corresponding to each M code is displayed.

- Setting a group number

To set an M code group number on the “M code group setting screen (Fig. 11.3 (a)),” use he following procedure:

1 Select the MDI mode.
2 Set “PARAMETER WRITE” on the setting screen to 1.
3 Display the “M code group setting screen”.

4 Move the cursor to the M code to be set using page keys and cursor keys. You can also enter the number of the M code to be set and press soft key [NO.SRH] to move the cursor to the M code.

5 Enter a group number and press soft key [INPUT] or the  key.

The valid range of M code group numbers is from 1 to 127 (127 groups). If a value of 0 is input, it is not registered as an M code group.

- Examples of setting parameters Nos. 3441 to 3444

In the following examples, the number of digits of an M code is 4.
<1> to <4> indicate parameters Nos. 3441 to 3444.

(1) When <1> = 300, <2> = 400, <3> = 500, and <4> = 900 are set

Number
0000
: 100 codes
0099
0300
: 100 codes
0399
0400
: 100 codes
0499
0500
: 100 codes
0599
0900
: 100 codes
0999

M code groups can be set for M0000 to M0099, M0300 to M0599, and M0900 to M999.
M codes M0300 to M0599 and M0900 to M999 are added to the M code group setting screen.

(2) When <1> = 200, <2> = 0, <3> = 550, and <4> = 800 are set

Number
0000
:
0099
0200
:
0299
0550
:
0649
0800
:
0899

M code groups can be set for M0000 to M0099, M0200 to M0299, M0550 to M0649, and M0800 to M0899.
(The setting of parameter <2> is invalid because it is 0.)
In this case, M codes M0200 to M0299, M0550 to M0649, and M0800 to M0899 are added to the M code group setting screen.

11.3.2 Setting an M Code Group Number Using a Program

You can execute a program to set an M code group number and M code name. The command format is shown below.

Format

G10 L40 Pn Rg ;

Pn: “n” specifies an M code.
Rg: “g” specifies an M code group number.

NOTE
1 If the format is invalid, an alarm PS1144, “G10 FORMAT ERROR” is issued.
2 If an M code group cannot be set for the M code specified for the P command or if the group number specified for the R command is not within the range between 0 and 127, an alarm PS1305, “DATA OUT OF RANGE” is issued.

(Example)
Executing the following program sets M code group “1” to M03:
G10 L40 P03 R1 ;

11.3.3 M Code Group Check Function

When multiple M commands in a single block (enabled when bit 7 (M3B) of parameter No. 3404 is set to 1) are used, you can check the following items. You can also select whether to check the items using bit 1 (MGC) of parameter No. 3400.

(1) M code to be specified in a single block containing no other M codes
If an M code which must be specified in a single block containing no other M codes is specified together with another M code, an alarm PS5016, “ILLEGAL COMBINATION OF M CODES” is issued.
(2) M codes in the same group
If multiple M codes in the same group are specified together, an alarm PS5016 is issued.

The valid range of group numbers is from 0 to 127 (128 groups).
Group numbers 0 and 1 have special meaning. Note the following points:

- Each M code with group number 1 is assumed to be an M code to be specified in a single block containing no other M codes.
- For each M code with group number 0, the “same group M code check” is ignored. That is, when multiple M codes with group number 0 are specified in a single block, the alarm is not issued.
- For each M code with group number 0, the “check for an M code to be specified in a single block containing no other M codes” is not ignored. That is, if an M code with group number 1 and an M code with group number 0 are specified in a single block, the alarm is issued.
- For M codes that are not to be output to the machine such as M98, M99, M198, and M codes for subprogram and macro calls (set in parameters Nos. 6071 to 6079 and Nos. 6080 to 6089 and with the macro executor), be sure to set 0 as the group number.
- For M00, M01, M02, M30, and M codes for which buffering is suppressed (set in parameters Nos. 3411 to 3432 and No.11290 to 11299), be sure to set 1 as the group number.

11.4 SECOND AUXILIARY FUNCTIONS (B CODES)

Overview

If a value with a maximum of eight digits is specified after address B, the code signal and strobe signal are transferred for calculation of the rotation axis. The code signal is retained until the next B code is specified.
Only one B code can be specified for each block. When the maximum number of digits are specified by parameter No.3033, an alarm is issued if the number of digits of a command exceeds the specified number.
In addition, the address used for specifying the second auxiliary function can be changed to an address other than address B (address A, C, U, V, or W) by setting parameter No.3460.
However, the address used for the second auxiliary function cannot also be used as the address of the controlled axis. For details, refer to the manual available from the machine tool builder.

Explanation

- Range of specification

-99999999 to 99999999

- Output value

The value specified after the address of the second auxiliary function is output on the code signals B00 to B31. Note the following about a output value.

1. When a command with a decimal point or a negative command is disabled
(When bit 0 (AUP) of parameter No.3450 is set to 0)

   When the second auxiliary function with no decimal point is specified, the specified value is output on the code signals as is, regardless of the desktop calculator decimal point setting (bit 0 (DPI) of parameter No.3401).
   Example:

Specified value	Output value
B10	10

   When the second auxiliary function with a decimal point is specified, alarm PS0007, "ILLEGAL USE OF DECIMAL POINT" is issued.
   When the second auxiliary function is specified with a negative value, alarm PS0006, "ILLEGAL USE OF MINUS SIGN" is issued.

2. When a command with a decimal point or a negative command is enabled
(When bit 0 (AUP) of parameter No.3450 is set to 1)

   When the desktop calculator decimal point setting is not specified (when bit 0 (DPI) of parameter No.3401 is set to 0), if the second auxiliary function with no decimal point is specified, the specified value is output on the code signals as is.
   Example:

Specified value	Output value
B10	10

   When desktop calculator decimal point input is specified (when bit 0 (DPI) of parameter No.3401 is set to 1), if the second auxiliary function with no decimal point is specified, the specified value multiplied by a magnification is output on the code signals. (Magnifications are shown in Table 11.4 (a).)
   Example:

Specified value	Output value
B10	10000 (When metric input is used and the reference axis is IS-B. The magnification is 1000.)

   When the second auxiliary function with a decimal point is specified, the specified value multiplied by a magnification is output to the code signals. (Magnifications are shown in Table 11.4 (a).)
   Example:

Specified value	Output value
B10.	10000 (When metric input is used and the reference axis is IS-B. The magnification is 1000.)
B0.123	1230 (When inch input is used, the reference axis is IS-B, and parameter AUX is set to 1. The magnification is 10000.)

   The magnification is determined as shown below according to the setting unit of the reference axis (specified by parameter No.1031) and bit 0 (AUX) of parameter No.3405.

Table 11.4 (a) Magnifications for an output value when the second auxiliary function with a decimal point is specified for desktop calculator decimal point input

Setting unit		Bit 0 (AUX) of parameter No.3405 = 0	Bit 0 (AUX) of parameter No.3405 = 1
Metric input system	Reference axis: IS-A	100×	100×
	Reference axis: IS-B	1000×	1000×
	Reference axis: IS-C	10000×	10000×
	Reference axis: IS-D	100000×	100000×
	Reference axis: IS-E	1000000×	1000000×
Inch input system	Reference axis: IS-A	100×	1000×
	Reference axis: IS-B	1000×	10000×
	Reference axis: IS-C	10000×	100000×
	Reference axis: IS-D	100000×	1000000×
	Reference axis: IS-E	1000000×	10000000×

⚠ CAUTION

If a decimal fraction remains after multiplying the specified value with a decimal point by a magnitude in Table 11.4 (a), the fraction is truncated.

Example:

Specified value	Output value
B0.12345	1234 (When inch input is used, the reference axis is IS-B, and parameter AUX is set to 1. The magnification is 10000.)

NOTE

If the number of digits of the specified value exceeds the allowable number of digits (set by parameter No. 3033), alarm PS0003, “TOO MANY DIGIT” is issued.

When the specified value is multiplied by a magnitude in Table 11.4 (a), the allowable number of digits must be set for the resultant value.

Limitation

Addresses used for the second auxiliary functions (addresses specified with B or parameter No. 3460) cannot be used as the addresses used for controlled axis names.

12 PROGRAM MANAGEMENT

Chapter 12, "PROGRAM MANAGEMENT", consists of the following sections:

12.1 FOLDERS.....255
12.2 PROGRAMS.....258
12.3 RELATION WITH CONVENTIONAL FUNCTIONS.....260

Operation around program management is described in "PROGRAM MANAGEMENT" in OPERATION.

12.1 FOLDERS

Overview

Folders can be created in program memory.

12.1.1 Folder Configuration

The following folders can be created:

- Folder names are up to 32 characters long.
- The following characters can be used in folder names:
  Alphabetical characters (uppercase and lowercase letters), numeric characters, and the symbols below:
  − + _ .
  Because "." and ".." are reserved folder names, they cannot be used.

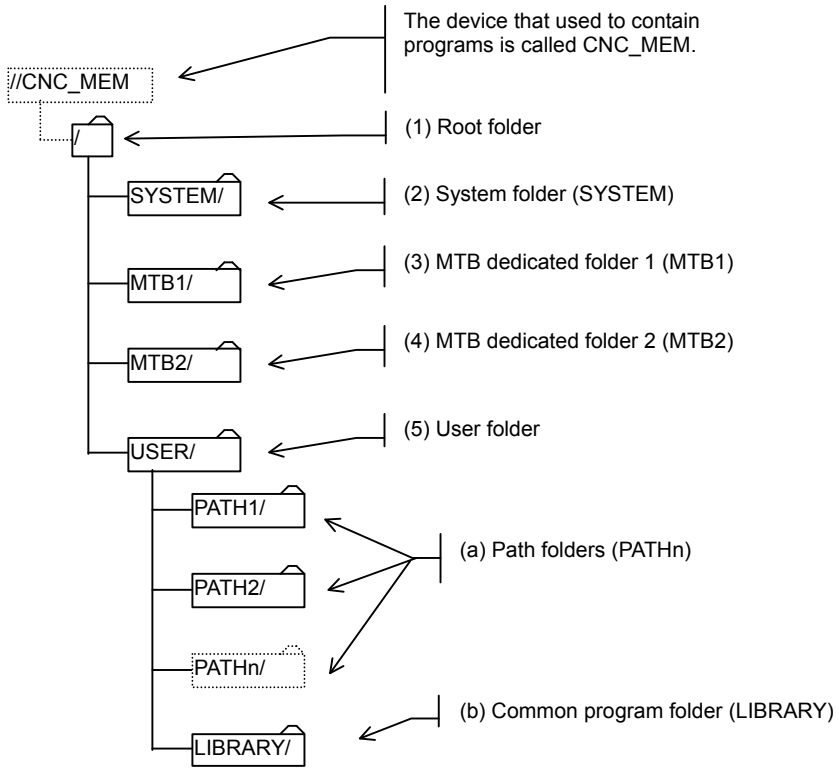
- Initial folders

When program memory is initialized, folders having predefined structures and names are created. These folders are called initial folders.

(1) Root folder
The parent folder of all folders
(2) System folder（SYSTEM）
Contains subprograms and macro programs of the system.
(3) MTB dedicated folder 1 (MTB1)
Contains subprograms and macro programs created by the machine tool builder.
(4) MTB dedicated folder 2 (MTB2)
Contains subprograms and macro programs created by the machine tool builder.
(5) User folder
Contains programs created by the user.
The following folders are further created in this folder:
(a) Path folders (PATHn: As many folders as the number of paths are created.)
Each contains main programs, subprograms, and macro programs used for the corresponding path.
(b) Common program folder (LIBRARY)
Contains subprograms and macro programs used commonly.

NOTE
The initial folders can neither be deleted nor renamed.

[Initial folder configuration]



- User created folders

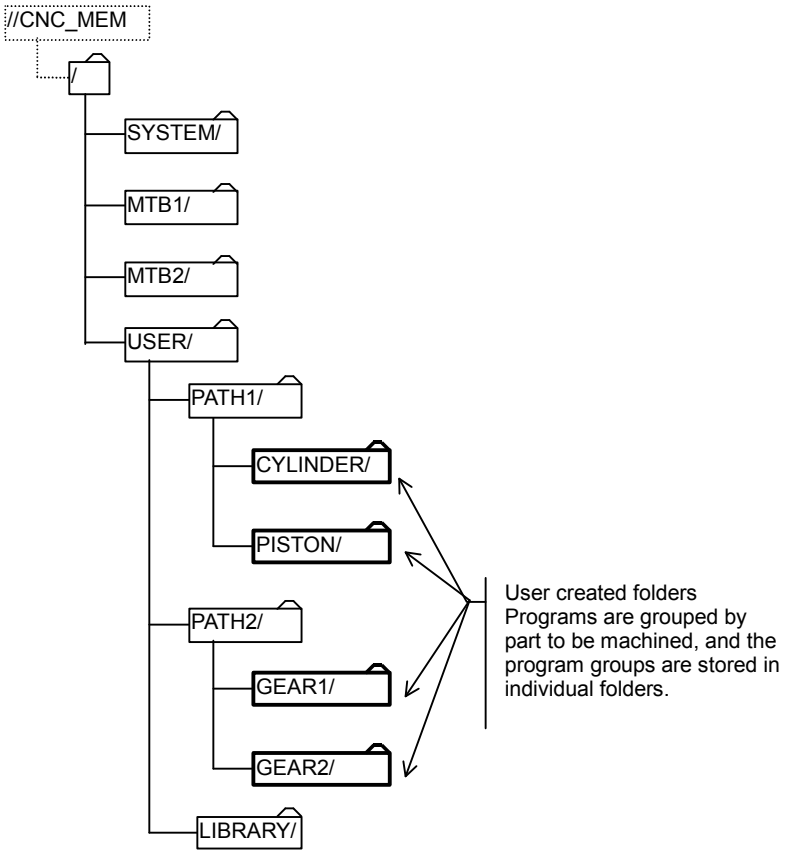
Folders other than the initial folders are called user created folders.
User created folders can be created in the following initial folders:

- User folder
- Path folders

User created folders can contain user created main programs, subprograms, and macro programs.

NOTE
1 Folder names must each be unique in the same folder.
2 Each time a user created folder is created, the number of programs that can be registered decreases by one.
3 Folder hierarchical levels of user created folders are limited.
Up to three hierarchical levels starting from the user folder (USER/) are permitted.

[Sample folder configuration]



12.1.2 Folder Attributes

The following attributes can be set for folders except the root folder:

- Edit disable
- Edit/display disable

- Edit disable

Editing of the programs and folders in a folder can be disabled.
A program in the folder can be output to an external device.
A program cannot be input (registered) from an external device to the folder.

- Edit/display disable

Editing and display of the programs and folders in a folder can be disabled.
When this attribute is set for a folder, the programs and folders in the folder become invisible. (The folder appears to contain nothing.)
A program in the folder cannot be output to an external device and a program cannot be input (registered) from an external device to a folder.

12.1.3 Default Folders

Default folders are folders on which operations are performed when no folder is specified. There are two types of default folders as follows:

- Foreground default folder
- Background default folder

How to select a default folder is described in "PROGRAM MANAGEMENT" in OPERATION.

- Foreground default folder

A folder used for foreground operations except automatic operations and program editing is set.
The target operations include:

- Program input/output
- External data input
- External workpiece number search

- Background default folder

A folder used for background operations is set.
The target operations include:

- Program input/output
- External I/O device control

NOTE
1 When the foreground or background default folder is not set, a path folder, which is an initial folder, is assumed.
2 Foreground and background default folder settings are stored in default folder setting files.
3 When a program file, program folder, or program folder management file is cleared, the default folder setting file is cleared at the same time.

12.2 PROGRAMS

Overview

Desired program names can be given to part programs in program memory.

12.2.1 Program Name

Program names can be set as follows:

- Program names are up to 32 characters long.
- The following characters can be used in program names:
  Alphabetical characters (uppercase and lowercase letters), numeric characters, and the symbols below:
  − + _ .
  Because "." and ".." are reserved program names, they cannot be used.

- Program names and program numbers

Program names are associated with program numbers as explained below.
When the program name of a program consists of "O" plus a numeric with the following limitation, the program can be handled also by program number.

- The numeric must be a leading zero suppressed value from 1 to 9999. (Bit 3 (ON8) of parameter No.11304 = 0)
- The numeric must be a leading zero suppressed value from 1 to 99999999. (Bit 3 (ON8) of parameter No.11304 = 0)

When the program name of a program does not have the above format, the program cannot be handled by program number.
When a program name consisting of "O" plus a numeric does not satisfy the above limitation, the program cannot be created.

Example)

Program names that can be treated as program numbers

O123	Program number 123
O1	Program number 1
O3000	Program number 3000
O9999	Program number 9999
O12345678	Program number 12345678

Program names that cannot be treated as program numbers

ABC
o123
O123.4

NOTE
1 Program names must each be unique in the same folder.
2 When the program name of a program is not treated as a program number, the program is restricted as follows:
- The program cannot be specified by program number.
- Information output by program number is impossible..

- Displaying Program Names and Program Numbers

The program name of the program selected or being executed as the main program is displayed as shown in Fig. 12.2.1 (a) to Fig. 12.2.1 (c).

- For program names that can be handled as program numbers, the program number is displayed.

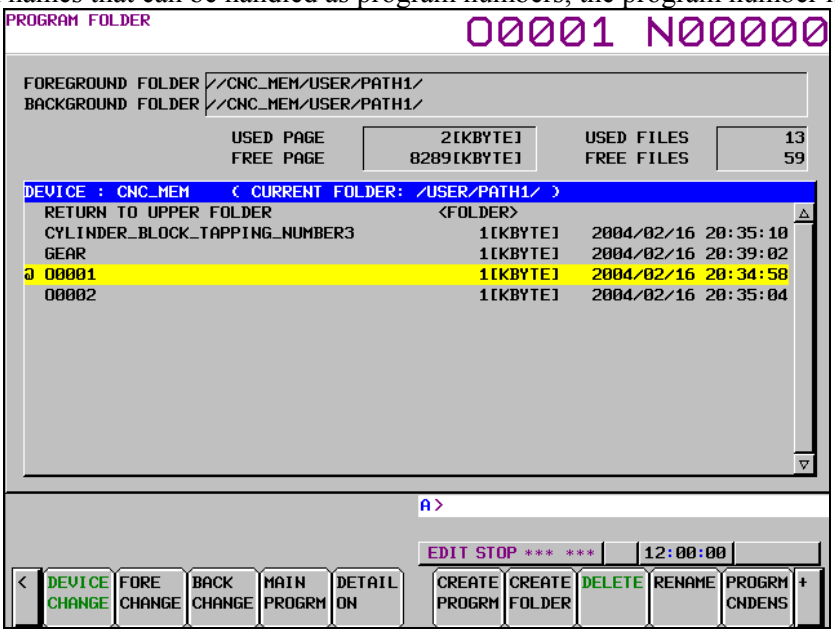


Fig. 12.2.1 (a) Program name display 1

- For program names that cannot be handled as program numbers, the program name is displayed. The display layout varies as shown below, depending on the length of a program name.

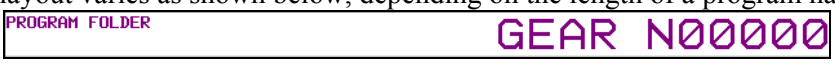


Fig. 12.2.1 (b) Program name display 2



Fig. 12.2.1 (c) Program name display 3

12.2.2 Program Attributes

The following attributes can be set for programs:

- Edit disable
- Edit/display disable
- Encoding
- Change protection level/output protection level

- Edit disable

Editing of a specified program can be disabled.
A program cannot be input (registered) from an external device to the folder.

- Edit/display disable

Editing and display of a specified program can be disabled.
When this attribute is set, the program becomes invisible.
(The folder appears not to contain this program.)
An output to an external device and an input from an external device (program registration) cannot be made.

- Encoding

A specified program can be encoded.
For details of encoding, see the description of the “Program encryption” function.
An output to an external device and an input from an external device (program registration) cannot be made.

- Change protection level/output protection level

With the 8-level data protection function, change and output protection can be provided for a specified program.
For details of the 8-level data protection function, see the description of the “Protection of data at eight levels” function.

12.3 RELATION WITH CONVENTIONAL FUNCTIONS

This section explains relation with conventional functions when folder names and program names are used.

12.3.1 Relation with Folders

This subsection explains how folders are used for operations and editing.

- Automatic operation
Main program

As the main program running for an automatic operation, a program in a desired folder can be selected.

Subprogram (called by M98/G72.1/G72.2)
Macro program (called by G65/G66/G66.1/M96)

- Subprogram call (M98)
- Macro call (simple call G65/modal call G66, G66.1)
- Macro interrupt (M96)
- Figure copying (G72.1, G72.2)

When a call listed above is made, folders are searched in the following order, and the program found first is called:

<1> Folders containing the main program
<2> Common program folder, which is an initial folder (LIBRARY)

Bit 7 (SCF) of parameter No. 3457 can be used to add the following search folders. (The folders are searched in the order listed below.) The folder to be actually enabled is separately set by bits 1 to 3 of parameters No. 3457.
<3> MTB dedicated folder 2, which is an initial folder (MTB2)
<4> MTB dedicated folder 1, which is an initial folder (MTB1)
<5> System folder, which is an initial folder (SYSTEM)

Subprogram (called by M code/specific address/2nd auxiliary function)
Macro program (called by G code/M code/T code/one touch macro)

- Subprogram call by M code/specific address/2nd auxiliary function code
- Macro call by G code/M code/T code
- One touch macro call

For programs called as listed above, the folders to be searched are set in bits 0 to 3 of parameter No.3457 in advance. (The search order is described below.)
The folders set as the search targets are searched, and the program found first is called.
<1> Common program folder, which is an initial folder (LIBRARY)
<2> MTB dedicated folder 2, which is an initial folder (MTB2)
<3> MTB dedicated folder 1, which is an initial folder (MTB1)
<4> System folder, which is an initial folder (SYSTEM)

- Program editing

A program in any folder can be edited.

- Program I/O

The following functions are performed for default folders:

- Program input from external devices
- Program output to external devices
  (Except the format with folder names)

The following functions are performed for a set background default folder:

- External I/O device control

- External data input

A set foreground default folder is searched for an external program.

- External workpiece number search

A set foreground default folder is searched for an external workpiece number.

- Macro executor

Programs called by execution macros, conversational macros, and auxiliary macros are programs in P-Code files regardless of the default folder.
For programs on the macro executor, the folder function and program name function are unavailable.

12.3.2 Relation with Program Names

Program names can be used with the following functions:

- Subprogram call (M98)
- Macro call (simple call G65/modal call G66, G66.1)
- Interruption type macro call (M96)

- Subprogram call in figure copying (G72.1, G72.2)
- Program I/O with external devices

- Subprogram call by program name
- Macro call by program name

- Subprogram call (M98)
- Macro call (G65/G66/G66.1)
- Interruption type macro call (M96)
- Figure copying (G72.1, G72.2)

When a program is called in the above functions, a subprogram call by program name and a macro call by program name can be used.

- Subprogram call by program name
  M98 <program-name> Lxx ;
- Macro call by program name
  G65 <program-name> Lxx argument-specification ;
  G66 <program-name> Lxx argument-specification ;
  G66.1 <program-name> Lxx argument-specification ;
- Interruption type macro call
  M96 <program-name> ;
- Subprogram call by program name in figure copying (G72.1, G72.2)
  G72.1 <program-name> Lxx Xxx Yxx Rxx ;
  G72.2 <program-name> Lxx Ixx Jxx ;

In the above subprogram and macro calls, the program having the program name indicated by <program-name> is called.

Format examples:

- Subprogram call
  M98 <R50> L1 ;
- Macro call
  G65 <R50> L1 A0 ;
  G66 <R50> L1 A1 ;
  G66.1 <R50> L1 A2 ;
- Interruption type macro call
  M96 <R50> ;
- Subprogram call in figure copy
  G72.1 <R50> L1 X0 Y0 R0 ;
  G72.2 <R50> L1 I0 J0 ;

NOTE
1 When characters in <> are read, they are treated in the same way as for characters in comments. So, note that these characters are treated differently from other significant information portions.
Refer to Appendix B “PROGRAM CODE LIST” for details.
2 The <program-name> word must be placed immediately after each word for calling (M98, G65, and so forth).

12.3.3 Related Parameters

This subsection lists the meanings of parameters related to program numbers and the folders and programs to be manipulated or executed.

Parameter No.	Bit No.	Description	Manipulation/execution target
3202	0 (NE8)	Disables or enables editing of programs O8000 to O8999.	Corresponding programs in all folders
	4 (NE9)	Disables or enables editing of programs O9000 to O9999.	Same as above
3204	3 (P8E)	Disables or enables editing of programs O80000000 to O8999999.	Same as above
	4 (P9E)	Disables or enables editing of programs O90000000 to O99999999.	Same as above
	5 (SPR)	Assumes or does not assume a particular program number in the nine thousands to be a number obtained by adding 90000000.	Same as above
3210/3211	-	Password/keyword for protecting programs in the nine thousands	Same as above
3222/3223	-	Program protection range (minimum value)/(maximum value)	Same as above
3404	2 (SBP)	In the subprogram call function, address P in the M198 block specifies a file/program number.	(Not dependent on the folder function and file name function)
6001	5 (TCS)	Calls or does not call a custom macro by T code.	The following initial folders can be selected by parameter setting: - Common program folder (LIBRARY) - MTB dedicated folder 2 (MTB2) - MTB dedicated folder 1 (MTB1) - System folder (SYSTEM)
6050 to 6059	-	G code for calling a custom macro with program No. 9010 to 9019	Same as above
6071 to 6079	-	M code for calling a subprogram with program No. 9001 to 9009	Same as above
6080 to 6089	-	M code for calling a custom macro with program No. 9020 to 9029	Same as above
6090/6091	-	ASCII code for calling a subprogram with program No. 9004/9005	Same as above
8341/8343	-	Target program number (for 4- and 8-digit O number) for sequence number comparison and stop	Foreground or background default folder depending on the mode
11304	3 (ON8)	Program number is 4-digit / 8-digit	Corresponding programs in all folders

12.3.4 Part Program Storage Size / Number of Registerable Programs

The following table lists the combinations of program storage sizes and the total number of registrable programs.

Part program storage size	Number of registerable programs	Number of registerable programs expansion 1	Number of registerable programs expansion 2 (*2)
32Kbyte (*1)	63	-	-
64Kbyte	63	125	-

Part program storage size	Number of registerable programs	Number of registerable programs expansion 1	Number of registerable programs expansion 2 (*2)
128Kbyte	63	250	-
256Kbyte	63	500	-
512Kbyte	63	1000	-
1Mbyte	63	1000	2000
2Mbyte	63	1000	4000
4Mbyte (*2)	63	1000	4000
8Mbyte (*2)	63	1000	4000

*1) For Series 32*i*-B only

*2) For Series 30*i*/31*i*-B, For Series 31*i*-B5 only

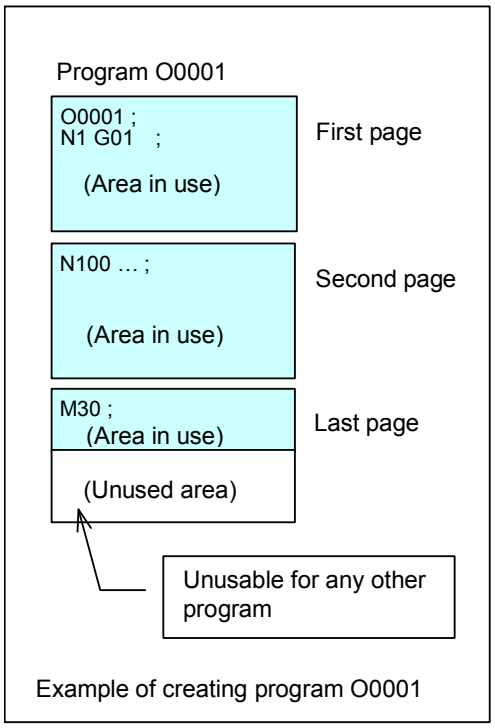


Fig. 12.3.4 (a) Unit of program storage

NOTE

1 Creating one folder results in the number of programs yet to be registerable decreasing one.

2 The program storage size means the maximum size of a program if the program is the one and only program registered.

3 If more than one program is registered, the total size of registerable programs reduces for the following reason.
The Series 30*i*/31*i*/32*i*-B and Series 31*i*-B5 manage programs in page units. The unit of program storage is managed also in page units. When a program is created, as many pages as necessary to store the program are secured, and the program is stored on these pages. Generally, the last program storage page has an unused area (Fig. 12.3.4 (a)). This unused area cannot be used to store any other program. For the sake of program management, it is regarded as an area in use.
The Series 16*i*/18*i*/21*i* uses a similar way of management, but the unit of pages in it differs from that in the Series 30*i*/31*i*/32*i*-B and FS31*i*-B5. So, if more than one program is registered in the Series 30*i*/31*i*/32*i*-B or FS31*i*-B5, the total program size of registerable programs in the Series 30*i*/31*i*/32*i*-B or Series 31*i*-B5 differs from that in the Series 16*i*/18*i*/21*i*.

13 PROGRAM CONFIGURATION

Overview

- Main program and subprogram

There are two program types, main program and subprogram. Normally, the CNC operates according to the main program. However, when a command calling a subprogram is encountered in the main program, control is passed to the subprogram. When a command specifying a return to the main program is encountered in a subprogram, control is returned to the main program.

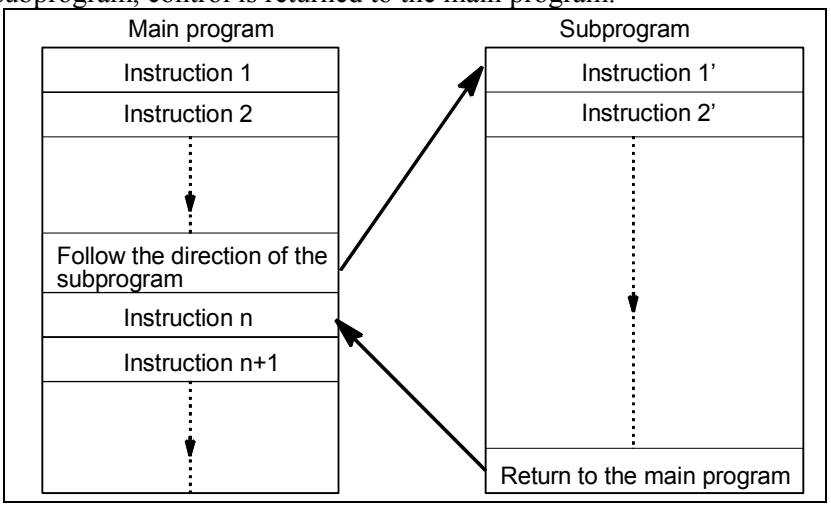


Fig. 13 (a) Main program and Subprogram

The CNC memory can hold up to 4000 main programs and subprograms (63 as standard). A main program can be selected from the stored main programs to operate the machine. See Chapter, "CREATING PROGRAMS" and Section, "PROGRAM SEARCH" for the methods of registering and selecting programs.

- Program components

A program consists of the following components:

Table 13 (a) Program components

Components	Descriptions
Program code start	Symbol indicating the start of a program file
Leader section	Used for the title of a program file, etc.
Program start	Symbol indicating the start of a program
Program section	Commands for machining
Comment section	Comments or directions for the operator
Program code end	Symbol indicating the end of a program file

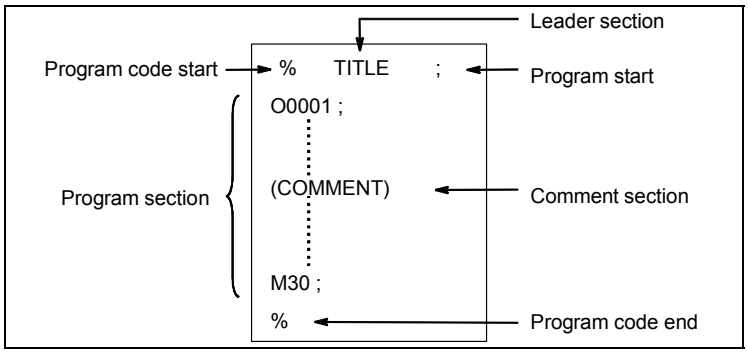


Fig. 13 (b) Program configuration

- Program section configuration

A program section consists of several blocks. A program section starts with a program number or program name and ends with a program end code.

Program section configuration	Program section
Program number	O0001 ;
Block 1	N1 G91 G00 X120.0 Y80.0 ;
Block 2	N2 G43 Z-32.0 H01 ;
:	:
Block n	Nn Z0 ;
Program end	M30 ;

A block contains information necessary for machining, such as a move command or coolant on/off command. Specifying a slash (/) at the start of a block disables the execution of some blocks (see "optional block skip" in II-13.2).

13.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS

This section describes program components other than program sections. See Section, "PROGRAM SECTION CONFIGURATION" for a program section.

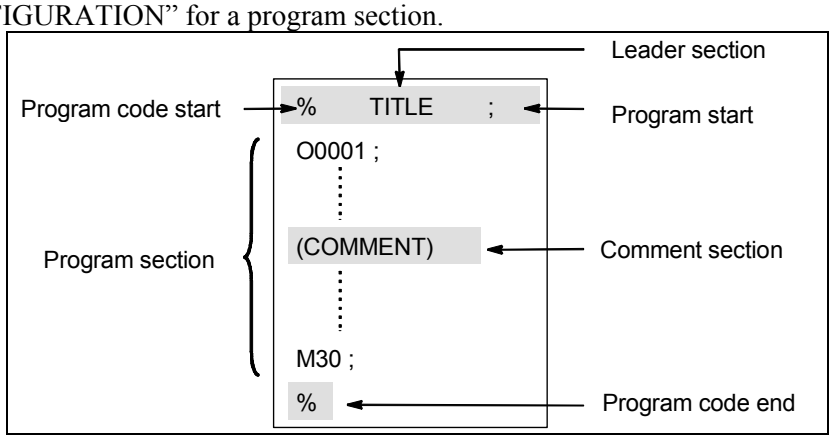


Fig. 13.1 (a) Program configuration

Explanation

- Program code start

The program code start indicates the start of a file that contains NC programs.
The mark is not required when programs are entered using SYSTEM P or ordinary personal computers. The mark is not displayed on the screen. However, if the file is output, the mark is automatically output at the start of the file.

Table 13.1 (a) Code of a program code start

Name	ISO code	EIA code	Notation in this manual
Program code start	%	ER	%

- Leader section

Data entered before the programs in a file constitutes a leader section.
When machining is started, the label skip state is usually set by turning on the power or resetting the system. In the label skip state, all information is ignored until the first end-of-block code is read. When a file is read into the CNC unit from an I/O device, leader sections are skipped by the label skip function.

A leader section generally contains information such as a file header. When a leader section is skipped, even a TV parity check is not made. So a leader section can contain any codes except the EOB code.

- Program start

The program start code is to be entered immediately after a leader section, that is, immediately before a program section.
This code indicates the start of a program, and is always required to disable the label skip function.
With SYSTEM P or ordinary personal computers, this code can be entered by pressing the return key.

Table 13.1 (b) Code of a program start

Name	ISO code	EIA code	Notation in this manual
Program start	LF	CR	;

NOTE
If one file contains multiple programs, the EOB code for label skip operation must not appear before a second or subsequent program number.

- Comment section

Any information enclosed by the control-out and control-in codes is regarded as a comment.
The user can enter a header, comments, directions to the operator, etc. in a comment section.

Table 13.1 (c) Codes of a control-in and a control-out

Name	ISO code	EIA code	Notation in this manual	Meaning
Control-out	(	2-4-5	(	Start of comment section
Control-in	)	2-4-7	)	End of comment section

When a program is read into memory for memory operation, comment sections, if any, are not ignored but are also read into memory. Note, however, that codes other than those listed in the code table in Appendix A are ignored, and thus are not read into memory.
When data in memory is output on external I/O device (See Chapter, “DATA INPUT/OUTPUT”), the comment sections are also output.
When a program is displayed on the screen, its comment sections are also displayed. However, those codes that were ignored when read into memory are not output or displayed.
During memory operation or DNC operation, all comment sections are ignored.
The TV check function can be used for a comment section by setting bit 1 (CTV) of parameter No. 0100.

⚠ CAUTION
If a long comment section appears in the middle of a program section, a move along an axis may be suspended for a long time because of such a comment section. So a comment section should be placed where movement suspension may occur or no movement is involved.

NOTE
1 If only a control-in code is read with no matching control-out code, the read control-in code is ignored.
2 The following codes cannot be used in the comment section:
- EOB
- % (ER for EIA)

- Program code end

A program code end is to be placed at the end of a file containing NC programs.
If programs are entered using the automatic programming system, the mark need not be entered.

The mark is not displayed on the screen. However, when a file is output, the mark is automatically output at the end of the file.
If an attempt is made to execute % when M02 or M30 is not placed at the end of the program, the alarm PS5010, “END OF RECORD” is occurred.

Table 13.1 (d) Code of a program code end

Name	ISO code	EIA code	Notation in this manual
Program code end	%	ER	%

13.2 PROGRAM SECTION CONFIGURATION

This section describes elements of a program section. See Section, “PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS” for program components other than program sections.

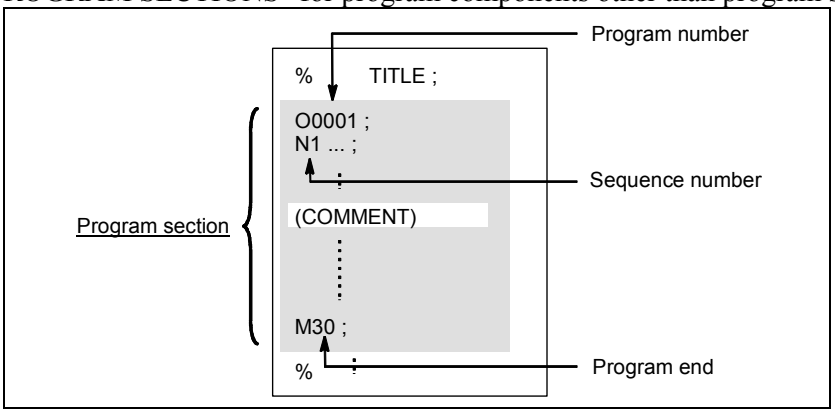


Fig. 13.2 (a) Program configuration

- Program number

A program number consisting of address O followed by a four-digit number is assigned to each program at the beginning registered in memory to identify the program. When the 8-digit number function is selected, the program number consists of eight digits.
In ISO code, the colon (:) can be used instead of O.
When no program number is specified at the start of a program, the sequence number (N....) at the start of the program is regarded as its program number. If a five-digit sequence number is used, the lower four digits are registered as a program number. If the lower four digits are all 0, the program number registered immediately before added to 1 is registered as a program number. Note, however, that N0 cannot be used for a program number.
If there is no program number or sequence number at the start of a program, a program number must be specified using the MDI panel when the program is stored in memory (See Section, “INPUT/OUTPUT ON EACH SCREEN” or “CREATING PROGRAMS USING THE MDI UNIT”)

NOTE
Program numbers 8000 to 9999 may be used by machine tool builders, and the user may not be able to use these numbers.

- Program name

A program name can be assigned instead of a program number.
When coding a program name, be sure to place the program name enclosed in "<" and ">" at the beginning of a program.

Example) % ;
<PARTS_1> ;
N1 ...
:
M30 ;
%

NOTE
A program name can be coded:
- At the beginning of a program
- Immediately after M98, G65, G66, G66.1, M96, G72.1, or G72.2

Do not code a file name in other than the above.

- Sequence number and block

A program consists of several commands. One command unit is called a block. One block is separated from another with an EOB of end of block code.

Table 13.2 (a) EOB code

Name	ISO code	EIA code	Notation in this manual
End of block (EOB)	LF	CR	;

At the head of a block, a sequence number consisting of address N followed by a number not longer than eight digits (1 to 99999999) can be placed. Sequence numbers can be specified in a random order, and any numbers can be skipped. Sequence numbers may be specified for all blocks or only for desired blocks of the program. In general, however, it is convenient to assign sequence numbers in ascending order in phase with the machining steps (for example, when a new tool is used by tool replacement, and machining proceeds to a new surface with table indexing.)

N300X200.0Z300.0 ; A sequence number is underlined.

Fig. 13.2 (b) Sequence number and block (example)

NOTE
N0 must not be used for the reason of file compatibility with other CNC systems. Program number 0 cannot be used. So 0 must not be used for a sequence number regarded as a program number.

- TV check (Vertical parity check)

A parity check is made for each block of input data. If the number of characters in one block (starting with the code immediately after an EOB and ending with the next EOB) is odd, an alarm PS0002, “TV ERROR” is output.
No TV check is made only for those parts that are skipped by the label skip function. Bit 1 (CTV) of parameter No. 0100 is used to specify whether comments enclosed in parentheses are counted as characters during TV check. The TV check function can be enabled or disabled by setting on the MDI unit (See Subsection, “Displaying and Entering Setting Data”).

- Block configuration (word and address)

A block consists of one or more words. A word consists of an address followed by a number some digits long. (The plus sign (+) or minus sign (-) may be prefixed to a number.)
For an address, one of the letters (A to Z) is used ; an address defines the meaning of a number that follows the address.

Word = Address + number (Example : X-1000)

Table 13.1 (b) indicates the usable addresses and their meanings.
The same address may have different meanings, depending on the preparatory function specification.

Table 13.2 (b) Major functions and addresses

Function	Address	Meaning
Program number	O(*)	Program number
Sequence number	N	Sequence number
Preparatory function	G	Specifies a motion mode (linear, arc, etc.)
Dimension word	X, Y, Z, U, V, W, A, B, C	Coordinate axis move command
	I, J, K	Coordinate of the arc center
	R	Arc radius
Feed function	F	Rate of feed per minute, Rate of feed per revolution
Spindle speed function	S	Spindle speed
Tool function	T	Tool number
Auxiliary function	M	On/off control on the machine tool
	B	Table indexing, etc.
Program number designation	P	Subprogram number
Number of repetitions	P, L	Number of subprogram repetitions
Parameter	P, Q	Canned cycle parameter

M

Offset number	D, H	Offset number
Dwell	P, X	Dwell time

T

Dwell	P, X, U	Dwell time

NOTE
(*) In ISO code, the colon (:) can also be used as the address of a program number.

N_	G_	X_ Y_	F_	S_	T_	M_	;
Sequence number	Preparatory function	Dimension word	Feed-function	Spindle speed function	Tool function	Auxiliary function	

Fig. 13.2 (c) 1 block (example)

- Major addresses and ranges of command values

Major addresses and the ranges of values specified for the addresses are shown below. Note that these figures represent limits on the CNC side, which are totally different from limits on the machine tool side. For example, the CNC allows a tool to traverse up to about 100 m (in millimeter input) along the X axis. However, an actual stroke along the X axis may be limited to 2 m for a specific machine tool.
Similarly, the CNC may be able to control a cutting feedrate of up to 240 m/min, but the machine tool may not allow more than 3 m/min. When developing a program, the user should carefully read the manuals of the machine tool as well as this manual to be familiar with the restrictions on programming.

Table 13.2 (c) Major addresses and ranges of command values

Function	Address	Input in mm	Input in inch
Program number	O (*1)	1 to 99999999	1 to 99999999
Sequence number	N	1 to 99999999	1 to 99999999
Preparatory function	G	0 to 99.9	0 to 99.9

Function		Address	Input in mm	Input in inch
Dimension word	Increment system IS-A	X,Y,Z,U,V, W,A,B,C,I, J,K,R (*2)	±999999.99 mm ±999999.99 deg.	±99999.999 inch *3 ±999999.99 deg.
	Increment system IS-B		±999999.999 mm ±999999.999 deg.	±99999.9999 inch *3 ±999999.999 deg.
	Increment system IS-C		±99999.9999 mm ±99999.9999 deg.	±9999.99999 inch *3 ±99999.9999 deg.
	Increment system IS-D		±9999.99999 mm ±9999.99999 deg.	±999.999999 inch *3 ±9999.99999 deg.
	Increment system IS-E		±999.999999 mm ±999.999999 deg.	±99.9999999 inch *3 ±999.999999 deg.
Feed per minute	Increment system IS-A	F	0.01 to 999000.00mm/min	0.001 to 96000.000inch/min
	Increment system IS-B		0.001 to 999000.000mm/min	0.0001 to 9600.0000inch/min
	Increment system IS-C		0.0001 to 99999.9999mm/min	0.00001 to 4000.00000inch/min
	Increment system IS-D		0.00001 to 9999.99999mm/min	0.000001 to 400.000000inch/min
	Increment system IS-E		0.000001 to 999.999999mm/min	0.0000001 to 40.0000000inch/min
Feed per revolution		F	0.001 to 50000mm/rev	0.0001 to 50.0000inch/rev
Spindle speed function		S (*4)	0 to 99999999	0 to 99999999
Tool function		T (*4)	0 to 99999999	0 to 99999999
Auxiliary function		M (*4)	0 to 99999999	0 to 99999999
		B (*4)	0 to 99999999	0 to 99999999
Offset number (M series only)		H, D	0 to 999	0 to 999
Dwell	Increment system IS-A	X, U (T series only)	0 to 999999.99 sec	0 to 999999.99 sec
	Increment system IS-B		0 to 99999.999 sec	0 to 99999.999 sec
	Increment system IS-C		0 to 9999.9999 sec	0 to 9999.9999 sec
	Increment system IS-D		0 to 999.99999 sec	0 to 999.99999 sec
	Increment system IS-E		0 to 99.999999 sec	0 to 99.999999 sec
Dwell		P	1 to 99999999	1 to 99999999
Designation of a program number		P	1 to 99999999	1 to 99999999
Number of subprogram repetitions		L	1 to 99999999	1 to 99999999
		P	0 to 9999	0 to 9999

*1 In ISO code, the colon (:) can also be used as the address of a program number.

*2 When address I, J, K, or R is used to specify the radius for circular interpolation, the specifiable range is as follows:

Increment system	Input in mm	Input in inch
IS-A	±999999999.99 mm	±99999999.999 inch
IS-B	±999999999.999 mm	±99999999.9999 inch
IS-C	±99999999.9999 mm	±9999999.99999 inch
IS-D	±9999999.99999 mm	±999999.999999 inch
IS-E	±999999.999999 mm	±99999.9999999 inch

*3 For inch input/millimeter machines, the maximum specifiable range of dimension words is as follows:

Increment system	Maximum specifiable range
IS-A	±39370.078 inch
IS-B	±39370.0787 inch
IS-C	±3937.00787 inch
IS-D	±393.700787 inch
IS-E	±39.3700787 inch

*4 The maximum value of addresses M, S, T, and B is 99999999. Note that, however, values longer than the permissible number of digits set in parameter No. 3030 to 3033 cannot be specified.

The values and uses for some codes are limited by parameter setting. (For example, some M codes are not buffered.) For details, refer to the parameter manual.

- Optional block skip

When a slash followed by a number (/n (n=1 to 9)) is specified at the head of a block, and optional block skip signals BDT1 to BDT9 are set to 1 during automatic operation, the information (/n to the end of the block (EOB)) contained in the block for which /n corresponding to signal BDTn is specified is ignored.

Example 1)
/2 N123 X100.0 Y200.0 ;
Example 2)
//3 N123 X100.0 Y200.0 ; → Incorrect
/1 /3 N123 X100.0 Y200.0 ; → Correct

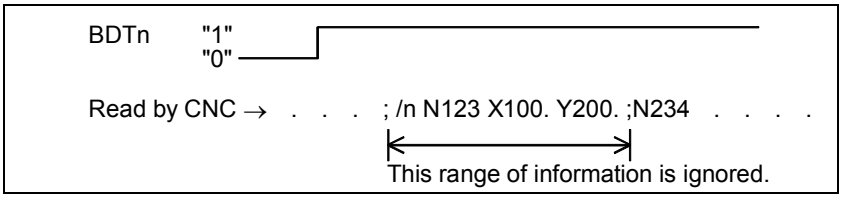
Input signal and program code

Input signal	Start code to be ignored
BDT1	/ or /1(NOTE)
BDT2	/2
BDT3	/3
BDT4	/4
BDT5	/5
BDT6	/6
BDT7	/7
BDT8	/8
BDT9	/9

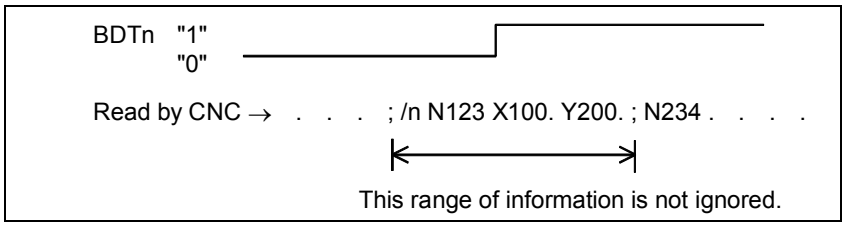
NOTE
1 Number 1 for /1 can be omitted. However, when two or more optional block skips are specified for one block, number 1 for /1 cannot be omitted.
2 Depending on the machine tool, all optional block skip signals (1 to 9) may not be usable. Refer to manuals of the machine tool builder to find which switches are usable.

The following shows the relationship between the timing at which optional block skip signals BDT1 to BDT9 are set to 1 and the range of information to be ignored.

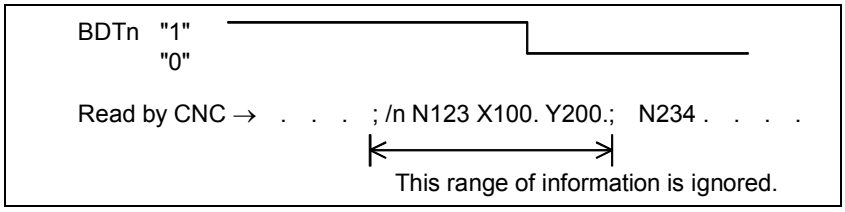
1. When the signal BDTn is set to 1 before the CNC starts reading a block that contains /n, the block is ignored.



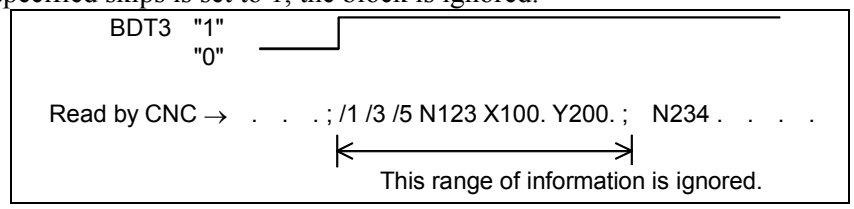
2. When the signal BDTn is set to 1 while the CNC is reading a block that contains /n, the block is not ignored.



3. When the signal BDTn is set to 0 while the CNC is reading a block that contains /n, the block is ignored.



4. Two or more optional block skips can be specified in one block. When the signal corresponding to any of the specified skips is set to 1, the block is ignored.



NOTE
1 This function is not used when a program is registered in memory. A block containing / is registered in memory regardless of the statuses of optional block skip signals. When a program in memory is also output regardless of the statuses of optional block skip signals.
In addition, the optional block skip function is enabled during a search for a sequence number.
2 Position of a slash
A block skip slash (/) must be specified at the beginning of a block. Note that if a slash is placed elsewhere, the information from the slash to immediately before the EOB code is ignored.
3 TV and TH check
When the optional block skip signal is set to 1, TH and TV checks are made for the skipped portions in the same way as when the optional block skip signal is set to 0.

- Program end

The end of a program is indicated by programming one of the following codes at the end of the program:

Table 13.2 (d) Code of a program end

Code	Meaning usage
M02	For main program
M03	
M99	For subprogram

If one of the program end codes is executed in program execution, the CNC terminates the execution of the program, and the reset state is set. When the subprogram end code is executed, control returns to the program that called the subprogram.

⚠ CAUTION
A block containing an optional block skip code such as /M02 ; , /M30 ; , or /M99 ; is not regarded as the end of a program. (see "Optional block skip".)

13.3 SUBPROGRAM (M98, M99)

If a program contains a fixed sequence or frequently repeated pattern, such a sequence or pattern can be stored as a subprogram in memory to simplify the program.
A subprogram can be called from the main program.
A called subprogram can also call another subprogram.

Format

- Subprogram configuration

One subprogram

```
Oxxxx ;
  :
M99;
```

Subprogram number or subprogram name (or the colon (:) optionally in the case of ISO)

Program end

M99 need not constitute a separate block as indicated below.

Example) X100.0Y100.0M99;

- Subprogram call

- When a subprogram with a 4-digit or shorter program number is called

M98 Pxxxx xxxx ;

Subprogram number

Number of times the subprogram is called repeatedly

or

M98 Pxxxx Lxxxxxxxx ;

Number of times the subprogram is called repeatedly

Subprogram number

- When a subprogram with a 5-digit or longer program number is called

M98 Pxxxxxxxx Lxxxxxxxx ;

Subprogram number

Number of times the subprogram is called repeatedly

- When a subprogram is called by program name

M98 <xxxx> Lxxxxxxxx ;

Subprogram name

Number of times the subprogram is called repeatedly

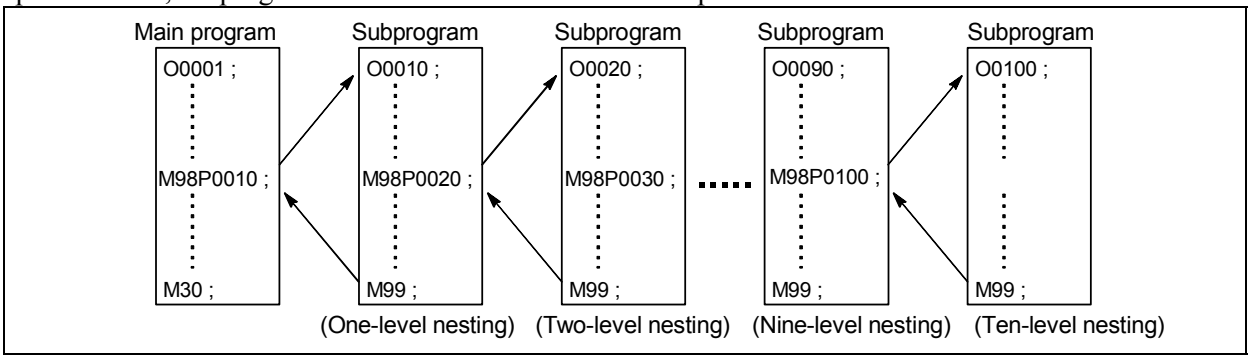
- Called program and folders to be searched

The order in which folders are searched depends on the method of calling a subprogram. Folders are searched in sequence and the program found first is called. For details, see the "Managing Programs" chapter.

NOTE
1 When calling a subprogram of a 4-digit or shorter subprogram number repeatedly (P8 digit), make the subprogram number 4 digits by prefixing it with "0" if the subprogram number is shorter than 4 digits.
Example)
P100100: Call subprogram No. 100 ten times.
P50001: Call subprogram No. 1 five times.
2 If the number of times the subprogram is called repeatedly is omitted when a subprogram with a 4-digit or shorter program number is called, the subprogram is called just once.
In this case, it is not necessary to adjust the subprogram number length to 4 digits as described in Item 1 above.
3 When calling a subprogram of a 4-digit or shorter program number repeatedly (P8 digit), do not specify address L in the same block.
4 When calling a subprogram with a 5-digit or longer program number, do not omit the specification of the number of repeats.
5 When calling a subprogram by program name, be sure to specify the program name immediately after M98.

Explanation

When the main program calls a subprogram, it is regarded as a one-level subprogram call. Thus, subprogram calls can be nested up to ten levels as shown below. In addition, when the custom macro option is used, subprogram and macro calls can be nested up to a total of 15 levels.



A single call command can repeatedly call a subprogram up to 99999999 times. For compatibility with automatic programming systems, in the first block, Nxxxxxxxx can be used instead of a subprogram number that follows O (or :). A sequence number after N is registered as a subprogram number.

NOTE
1 The M98 and M99 code signal and strobe signal are not output to the machine tool.
2 If the subprogram number specified by address P cannot be found, an alarm PS0078, "SEQUENCE NUMBER NOT FOUND" is output.
3 When an attempt is made to call a subprogram by program name, but the specified program cannot be found, alarm PS0310, "FILE NOT FOUND" is issued.

Example

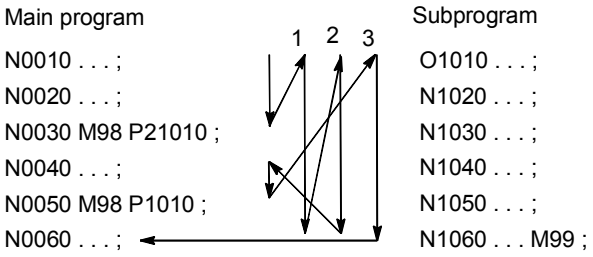
M98 P51002 ;

This command specifies "Call the subprogram (number 1002) five times in succession." A subprogram call command (M98P_) can be specified in the same block as a move command.

X1000.0 M98 P1200 ;

This example calls the subprogram (number 1200) after an X movement.

Execution sequence of subprograms called from a main program

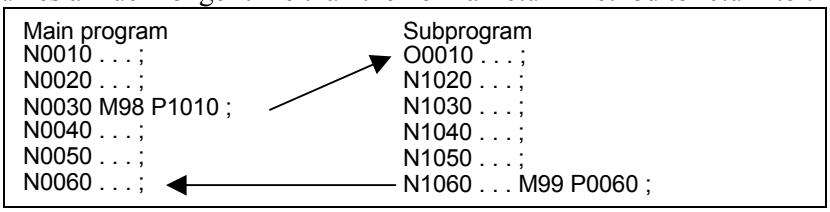


A subprogram can call another subprogram in the same way as a main program calls a subprogram.

Special usage

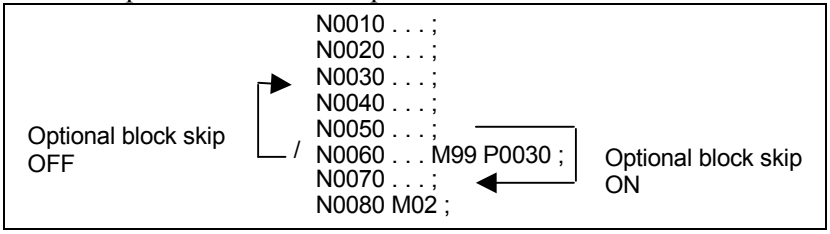
- Specifying the sequence number for the return destination in the main program

If P is used to specify a sequence number when a subprogram is terminated, control does not return to the block after the calling block, but returns to the block with the sequence number specified by P. Note, however, that P is ignored if the main program is operating in a mode other than memory operation mode. This method consumes a much longer time than the normal return method to return to the main program.



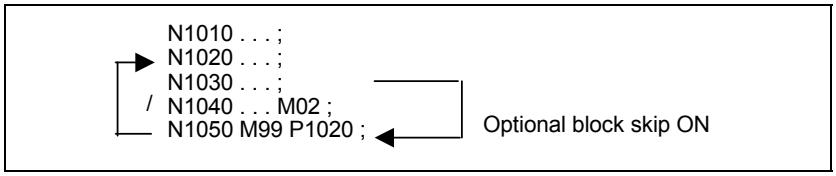
- Using M99 in the main program

If M99 is executed in a main program, control returns to the start of the main program. For example, M99 can be executed by placing /M99 ; at an appropriate location of the main program and setting the optional block skip function to off when executing the main program. When M99 is executed, control returns to the start of the main program, then execution is repeated starting at the head of the main program.
Execution is repeated while the optional block skip function is set to off.
If the optional block skip function is set to on, the /M99 ; block is skipped ; control is passed to the next block for continued execution.
If/M99Pn ; is specified, control returns not to the start of the main program, but to sequence number n. In this case, a longer time is required to return to sequence number n.



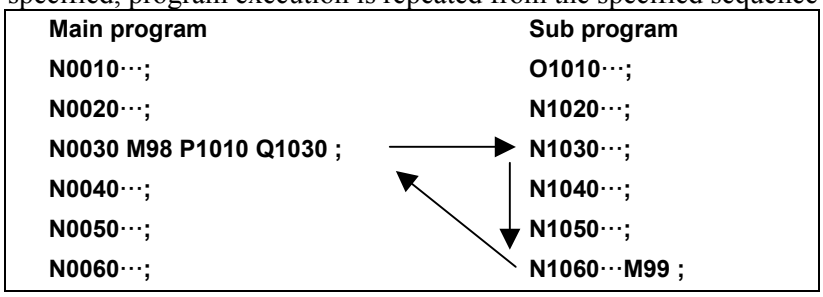
- Using a subprogram only

A subprogram can be executed just like a main program by searching for the start of the subprogram with the MDI.
(See Section, "PROGRAM SEARCH" for information about search operation.)
In this case, if a block containing M99 is executed, control returns to the start of the subprogram for repeated execution. If a block containing M99Pn is executed, control returns to the block with sequence number n in the subprogram for repeated execution. To terminate this program, a block containing /M02 ; or /M30 ; must be placed at an appropriate location, and the optional block switch must be set to off ; this switch is to be set to on first.

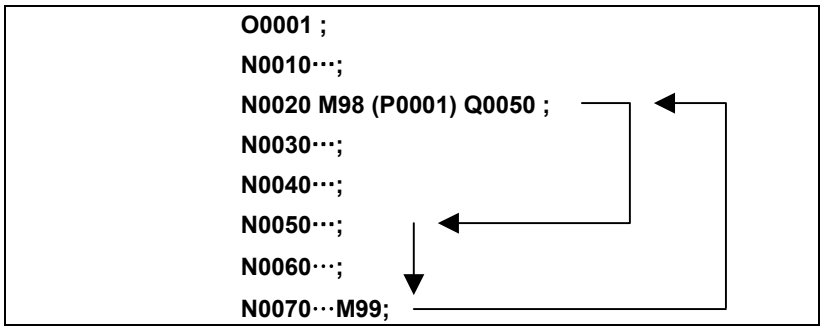


- Subprogram call with sequence number

Setting bit 0 (SQC) of parameter No. 6005 to 1 can call a specified sequence number in the subprogram for execution.
In a subprogram call command, specify the letter Q followed by a sequence number to be called after the letter P for specifying a program number.
M98 Pxxxx Qxxxxx ;
Sequence number
Program number
This command causes program execution to start at the called sequence number in the subprogram. If a repetition count is specified, program execution is repeated from the specified sequence number.



This function enables a sequence number in the same program to be called for execution as shown below. This method, however, requests the programmer to be aware of an allowable call nesting level. If an attempt is made to exceed the allowable nesting level, alarm PS0077, "TOO MANY SUB,MACRO NESTING" is issued.



For a call within the same program, specification of Pxxxx in a block can be omitted when the block includes M98.

This function is usable only for subprogram calls by M98; it is unusable for non-M98 calls, such as macro calls or external subprogram calls based on M198.

NOTE
If bit 0 (SQC) of parameter No. 6005 is ”0”, and an M98 Pxxxx Qxxxxx command is specified, alarm PS0009, "IMPROPER NC-ADDRESS", is issued.

14 FUNCTIONS TO SIMPLIFY PROGRAMMING

Chapter 14, "FUNCTIONS TO SIMPLIFY PROGRAMMING", consists of the following sections:

14.1 FIGURE COPYING (G72.1, G72.2) 279
14.2 3-DIMENSIONAL COORDINATE SYSTEM CONVERSION 284

14.1 FIGURE COPYING (G72.1, G72.2)

Machining can be repeated after moving or rotating the figure using a subprogram.

Format

- Rotational copying

Xp-Yp plane (specified by G17) : **G72.1 P_ L_ Xp_Yp_R_ ;**
Zp-Xp plane (specified by G18) : **G72.1 P_ L_ Zp_Xp_R_ ;**
Yp-Zp plane (specified by G19) : **G72.1 P_ L_ Yp_Zp_R _;**

P :Subprogram number
L :Number of times the operation is repeated
Xp :Center of rotation on the Xp axis (Xp: X-axis or an axis parallel to the X-axis)
Yp :Center of rotation on the Yp axis (Yp: Y-axis or an axis parallel to the Y-axis)
Zp :Center of rotation on the Zp axis (Zp: Z-axis or an axis parallel to the Z-axis)
R :Angular displacement (A positive value indicates a counterclockwise angular displacement. Specify an incremental value.)

Specify a plane selection command (G17, G18, or G19) to select the plane on which the rotational copying is made.

- Linear copying

Xp-Yp plane (specified by G17) : **G72.2 P_ L_ I_ J_ ;**
Zp-Xp plane (specified by G18) : **G72.2 P_ L_ K_ I_ ;**
Yp-Zp plane (specified by G19) : **G72.2 P_ L_ J_ K_;**

P :Subprogram number
L :Number of times the operation is repeated
I :Shift along the Xp axis
J :Shift along the Yp axis
K :Shift along the Zp axis

Specify a plane selection command (G17, G18, or G19) to select the plane on which the linear copying is made.

Explanation

- First block of the subprogram

Always specify a move command in the first block of a subprogram that performs a rotational or linear copying. If the first block contains only the program number such as O1234; and does not have a move command, movement may stop at the start point of the figure made by the n-th (n = 1, 2, 3, ...) copying.
Specify the first move command in the absolute mode.
(Example of an incorrect program)

```
O1234 ;
G00 G90 X100.0 Y200.0 ;
. . . . . ;
. . . . . ;
M99 ;
```

(Example of a correct program)

```
O1000 G00 G90 X100.0 Y200.0 ;
. . . . . ;
. . . . . ;
M99 ;
```

- Combination of rotational and linear copying

The linear copying command can be specified in a subprogram for a rotational copying. Also, the rotational copying command can be specified in a subprogram for a linear copying.

- Subprogram call

In a subprogram for rotational or linear copying, M98 for calling another subprogram or G65 for calling a macro can be specified.

- Specifying the center of rotation

The center of rotation specified with G72.1 is processed as an absolute position even in the incremental mode.

- Specifying address

In a block with G72.1, addresses other than P, L, Xp, Yp, Zp, or R are ignored. The subprogram number (P), coordinates of the center of rotation (Xp, Yp, Zp), and angular displacement (R) must be specified.
In a block with G72.2, addresses other than P, L, I, J, or K are ignored.
The subprogram number (P) and shift (I, J, K) must be specified.

- Address P

If the subprogram number specified with P is not found, alarm PS0078, “SEQUENCE NUMBER NOT FOUND” occurs. If P is not specified, alarm PS0076, “PROGRAM NOT FOUND” occurs.

- Address L

If L is omitted, the repetition count is assumed to be 1 and the subprogram is called only once.

- Increment in angular displacement or shift

In a block with G72.1, an increment in angular displacement is specified with address R. The angular displacement of the figure made by the n-th rotation is calculated as follows : $R \times (n - 1)$.
In a block with G72.2, an increment in shift is specified with addresses I, J, and K. The shift of the figure made by the n-th movement is calculated as follows : (Programmed shift) $\times (n - 1)$.

- Nesting level of a subprogram

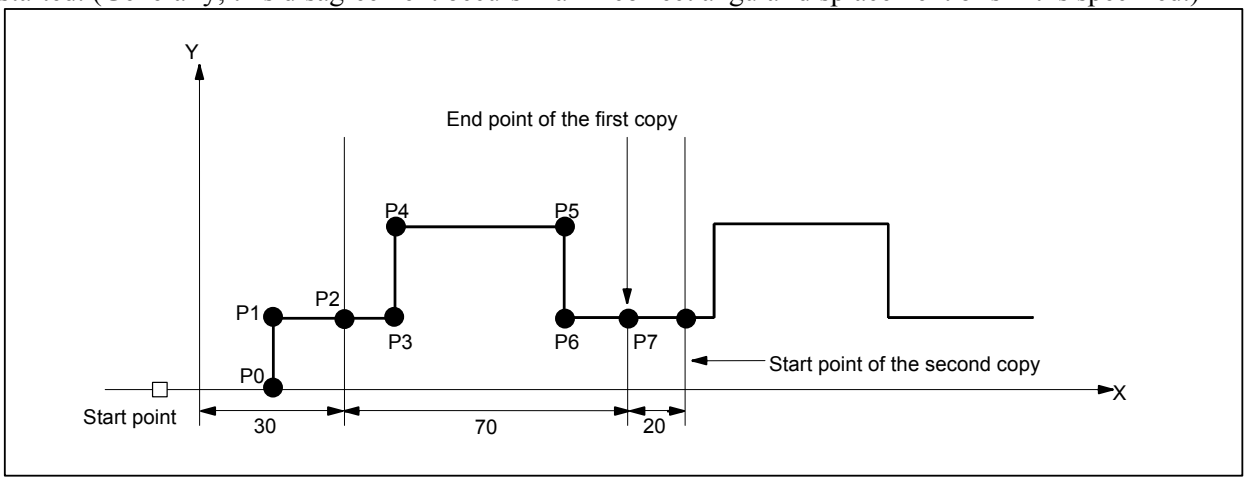
If a subprogram is called by G72.1 or G72.2, the nesting level is increased by one in the same manner as when M98 is specified.

- Block end position

The coordinates of a figure moved rotationally or linearly (block end position) can be read from #5001 and subsequent system variables of the custom macro of rotational or linear copying.

- Disagreement between end point and start point

If the end point of the figure made by the n-th copy does not agree with the start point of the figure to be made by the next (n + 1) copy, the figure is moved from the end point to the start point, then copying is started. (Generally, this disagreement occurs if an incorrect angular displacement or shift is specified.)




```
Main program
O1000 ;
N10 G92 X-20.0 Y0.0 ;
N20 G00 G90 X0.0 Y0.0 ;
N30 G01 X20.0 Y0.0 F10 ;          (P0)
N40 Y20.0 ;                       (P1)
N50 X30.0 ;                       (P2)
N60 G72.2 P2000 L3 I90.0 J0.0 ;
                    └──  Although a shift of 70 mm was required, I90.0 was
                         specified instead of I70.0. Since an incorrect shift was
                         specified, the end point of the figure made by the n-th
                         copy disagrees with the start point of the figure to be
                         made by the next (n + 1) copy.

Subprogram
O2000 G90 G01 X40.0 ;             (P3)
N100 Y40.0 ;                      (P4)
N200 X80.0 ;                      P5)
N300Y20.0 ;                       (P6)
N400 X100.0 ;                     (P7)
N500 M99;
```

Limitation

- Specifying two or more commands to copy a figure

G72.1 cannot be specified more than once in a subprogram for making a rotational copying (If this is attempted, alarm PS0160, “MISMATCH WAITING M-CODE” will occur). G72.2 cannot be specified more than once in a subprogram for making a linear copying (If this is attempted, alarm PS0161, “ILLEGAL P OF WAITING M-CODE” will occur).

- Commands that must not be specified

Within a program that performs a rotational or linear copying, the following must not be specified:

- Command for changing the selected plane (G17 to G19)
- Command for specifying polar coordinates
- Reference position return command
- Coordinate system rotation, scaling, programmable mirror image
  The command for rotational or linear copying can be specified after a command for coordinate system rotation, scaling, or programmable mirror image is executed.

- Modes that must not be selected

The figure cannot be copied during chamfering, corner rounding, or tool offset.

- Unit system

The two axes of the plane for copying a figure must have an identical unit system.

- Single block

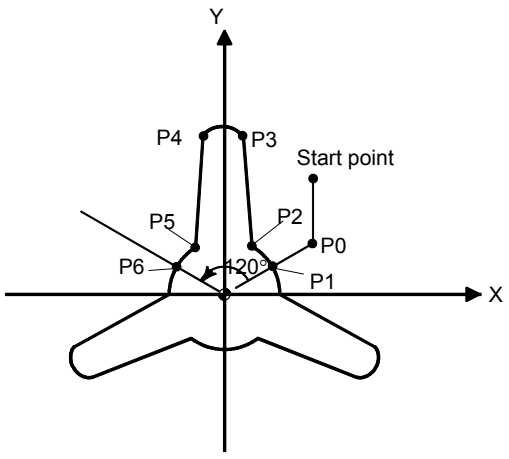
Single-block stops are not performed in a block with G72.1 or G72.2.

- Specifying tool radius compensation and the workpiece coordinate system

In a subprogram for copying a figure, the G code for tool radius / tool nose radius compensation or compensation amount (H or D code) cannot be changed. G92 and G54 to G59 cannot be changed either. Those codes must be specified before figure copying is started.

Example

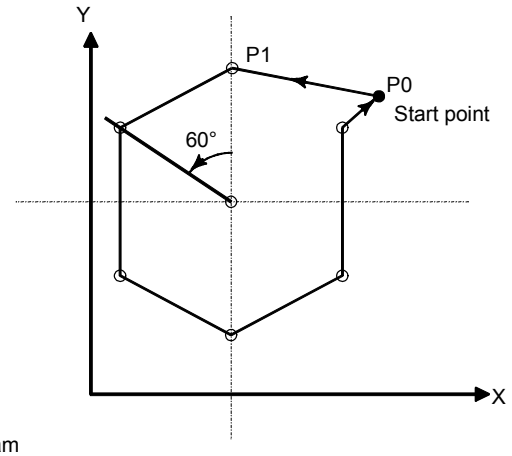
- Rotational copying




```
Main program
  O1000 ;
  N10 G92 X40.0 Y50.0 ;
  N20 G00 G90 X_ Y_ ;                  (P0)
  N30 G01 G17 G41 X_ Y_ D01 F10 ;      (P1)
  N40 G72.1 P2000 L3 X0 Y0 R120.0 ;
  N50 G40 G01 X_ Y_ I_ J_ ;            (P0)
  N60 G00 X40.0 Y50.0 ;
  N70 M30 ;

Sub program
  O2000 G03 X_ Y_ R30.0 ;              (P2)
  N100 G01 X_ Y_ ;                     (P3)
  N200 G03 X_ Y_ R10.0 ;               (P4)
  N300 G01 X_ Y_ ;                     (P5)
  N400 G03 X_ Y_ R30.0 ;               (P6)
  N500 M99;
```

- Rotational copying (spot boring)



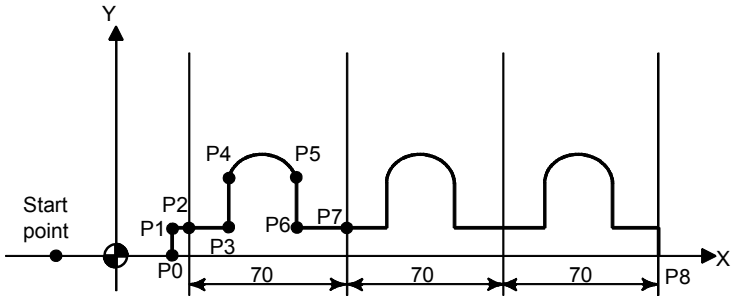
Main program

```
O3000 ;
N10 G92 G17 X80.0 Y50.0 ;                (P0)
N20 G72.1 P4000 L6 X0 Y0 R60.0 ;
N30 G80 G00 X80.0 Y50.0 ;                (P0)
N40 M30 ;
```

Subprogram

```
O4000 N100 G90 G81 X_ Y_ R_ Z_ F_ ;      (P1)
N200 M99 ;
```

- Linear copying



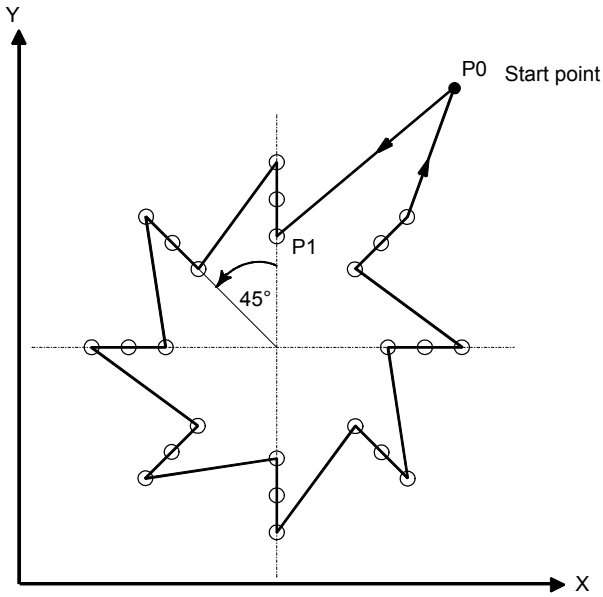
Main program

```
O1000 ;
N10 G92 X-20.0 Y0 ;
N20 G00 G90 X0 Y0 ;
N30 G01 G17 G41 X20. Y0 D01 F10 ;   (P0)
N40 Y20. ;                          (P1)
N50 X30. ;                          (P2)
N60 G72.2 P2000 L3 I70.0 J0 ;
N70 X_ Y_ ;                         (P8)
N80 X0 ;
N90 G00 G40 X-20.0 Y0 ;
N100 M30 ;
```

Subprogram

```
O2000 G90 G01 X_ ;                  (P3)
N100 Y_ ;                           (P4)
N200 G02 X_ I_ ;                    (P5)
N300 G01 Y_ ;                       (P6)
N400 X_ ;                           (P7)
N500 M99 ;
```

- Combination of rotational copying and linear copying (bolt hole circle)



Main program

```
O1000 ;
N10 G92 G17 X100.0 Y80.0 ;                (P0)
N20 G72.1 P2000 X0 Y0 L8 R45.0 ;
N30 G80 G00 X100.0 Y80.0 ;                (P0)
N40 M30 ;
```

Subprogram (rotational copy)

```
O2000 N100 G72.2 P3000 I0 J_ L3 ;
N200 M99 ;
```

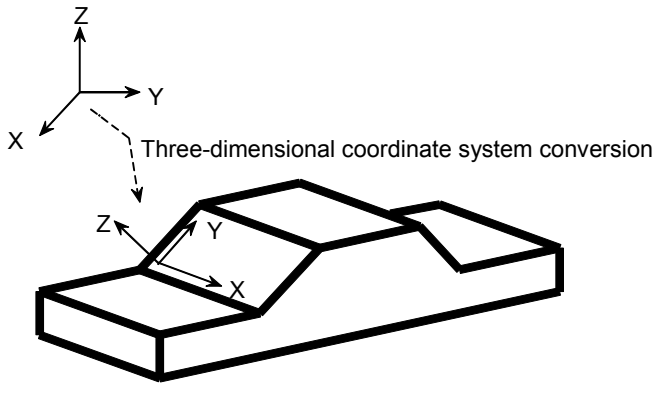
Subprogram (linear copy)

```
O3000 N110 G90 G81 X_ Y_ R_ Z_ F_ ;    (P1)
N210 M99 ;
```

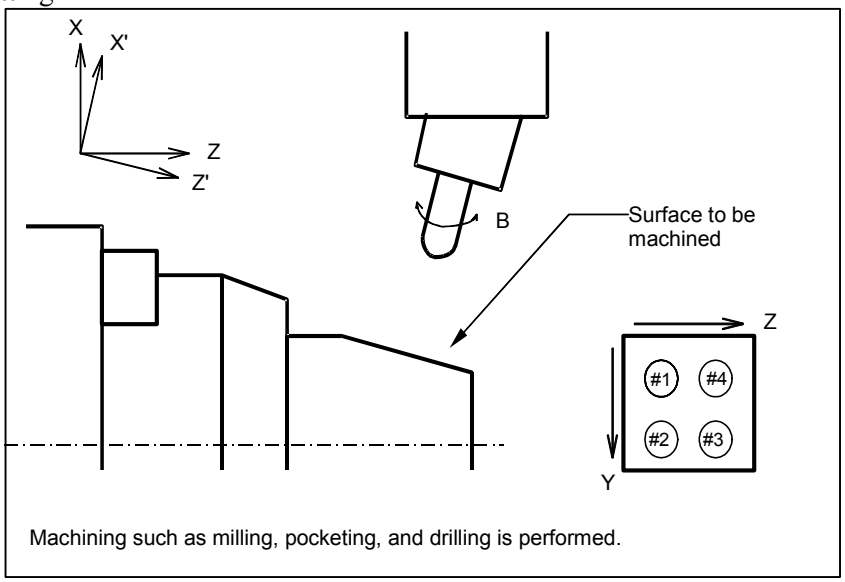
14.2 3-DIMENSIONAL COORDINATE SYSTEM CONVERSION

Coordinate system conversion about an axis can be carried out if the center of rotation, direction of the axis of rotation, and angular displacement are specified. This function is very useful in 3-dimensional machining by a die-sinking machine or similar machine. For example, if a program specifying machining on the XY plane is converted by the 3-dimensional coordinate system conversion function, the identical machining can be executed on a desired plane in 3-dimensional space.

- For milling machining



- For lathe cutting



Machining such as milling, pocketing, and drilling is performed.

Format

M

G68 Xp_{X1} Yp_{Y1} Zp_{Z1} I_{i1} J_{j1} K_{k1} Rα ; Starting 3-dimensional coordinate system conversion

:
: } 3-dimensional coordinate system conversion mode

G69 ; Canceling 3-dimensional coordinate system conversion

Xp, Yp, Zp : Center of rotation (absolute coordinates) on the X, Y, and Z axis or parallel axes
I, J, K : Direction of the axis of rotation
R : Angular displacement

T

G68.1 Xp_{X1} Yp_{Y1} Zp_{Z1} I_{i1} J_{j1} K_{k1} Rα ; Starting 3-dimensional coordinate system conversion

:
: } 3-dimensional coordinate system conversion mode

G69.1 ; Canceling 3-dimensional coordinate system conversion

Xp, Yp, Zp : Center of rotation (absolute coordinates) on the X, Y, and Z axis or parallel axes
I, J, K : Direction of the axis of rotation
R : Angular displacement

NOTE
The G code of this function is hereinafter described using the format (G68/G69) for the machining center system in this section.

Explanation

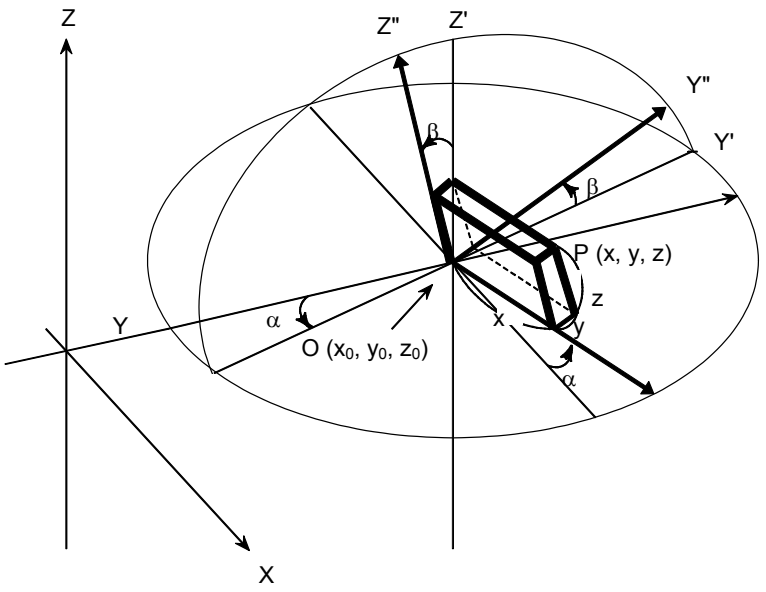
- Command for 3-dimensional coordinate system conversion (program coordinate system)

N1 G68 Xp x_1 Yp y_1 Zp z_1 I i_1 J j_1 K k_1 R α ;
N2 G68 Xp x_2 Yp y_2 Zp z_2 I i_2 J j_2 K k_2 R β ;
N3
:
Nn G69 ;

3-dimensional coordinate system conversion can be executed twice.
In the N1 block, specify the center, direction of the axis of rotation, and angular displacement of the first rotation.

When this block is executed, the center of the original coordinate system is shifted to (x_1, y_1, z_1), then rotated around the vector (i_1, j_1, k_1) by angular displacement α. The new coordinate system is called X'Y'Z'. In the N2 block, specify the center, direction of the axis of rotation, and angular displacement of the second rotation. In the N2 block, specify coordinates and the angle with the coordinate system formed after the N1 block in Xp, Yp, Zp, I, J, K, and R. When the N2 block is executed, the X'Y'Z' coordinate system is shifted to (x_2, y_2, z_2), then rotated around the vector (i_2, j_2, k_2) by angular displacement β. The newest coordinate system is called X"Y"Z". In the subsequent N3 block, coordinates in the X"Y"Z" coordinate system are specified with Xp, Yp, and Zp. The X"Y"Z" coordinate system is called the program coordinate system.
If (Xp, Yp, Zp) is not specified in the N2 block, (Xp, Yp, Zp) in the N1 block is assumed to be the center of the second rotation (the N1 and N2 blocks have a common center of rotation). If the coordinate system is to be rotated only once, the N2 block need not be specified.

Example) G68 Xx_0 Yy_0 Zz_0 I0 J0 K1 Rα ;
G68 I1 J0 K0 Rβ ;



X, Y, Z	:	Workpiece coordinate system
X', Y', Z'	:	Coordinate system formed after the first conversion
X", Y", Z"	:	Coordinate system formed after the second conversion
α	:	Angular displacement of the first rotation
β	:	Angular displacement of the second rotation
O (x_0, y_0, z_0)	:	Center of rotation
P (x, y, z)	:	Coordinates in the X"Y"Z" coordinate system (program coordinate system)

- Format error

If one of the following format errors is detected, alarm PS5044, “G68 FORMAT ERROR” occurs:

1. When I, J, or K is not specified in a block with G68
   (a parameter of coordinate system rotation is not specified)
2. When I, J, and K are all set to 0 in a block with G68
3. When R is not specified in a block with G68

- Center of rotation

Specify absolute coordinates with Xp, Yp, and Zp in the G68 block.

- Equation for 3-dimensional coordinate system conversion

The following equation shows the general relationship between (x, y, z) in the program coordinate system and (X, Y, Z) in the original coordinate system (workpiece coordinate system).

$$\begin{pmatrix} X \\ Y \\ Z \end{pmatrix} = (M_1)\begin{pmatrix} x \\ y \\ z \end{pmatrix} + \begin{pmatrix} x_1 \\ y_1 \\ z_1 \end{pmatrix}$$

When conversion is carried out twice, the relationship is expressed as follows:

$$\begin{pmatrix} X \\ Y \\ Z \end{pmatrix} = (M_1)(M_2)\begin{pmatrix} x \\ y \\ z \end{pmatrix} + (M_1)\begin{pmatrix} x_2 \\ y_2 \\ z_2 \end{pmatrix} + \begin{pmatrix} x_1 \\ y_1 \\ z_1 \end{pmatrix}$$

X, Y, Z : Coordinates in the original coordinate system (workpiece coordinate system)
x, y, z : Programmed value (coordinates in the program coordinate system)
x_1, y_1, z_1 : Center of rotation of the first conversion
x_2, y_2, z_2 : Center of rotation of the second conversion
(coordinates in the coordinate system formed after the first conversion)
M_1 : First conversion matrix
M_2 : Second conversion matrix

M_1 and M_2 are conversion matrices determined by an angular displacement and rotation axis. Generally, the matrices are expressed as shown below.

$$\begin{pmatrix} {n_1}^2 + (1-{n_1}^2)\cos\theta & n_1 n_2(1-\cos\theta) - n_3\sin\theta & n_1 n_3(1-\cos\theta) + n_2\sin\theta \\ n_1 n_2(1-\cos\theta) + n_3\sin\theta & {n_2}^2 + (1-{n_2}^2)\cos\theta & n_2 n_3(1-\cos\theta) - n_1\sin\theta \\ n_1 n_3(1-\cos\theta) - n_2\sin\theta & n_2 n_3(1-\cos\theta) + n_1\sin\theta & {n_3}^2 + (1-{n_3}^2)\cos\theta \end{pmatrix}$$

n_1 : Cosine of the angle made by the rotation axis and X-axis i/p
n_2 : Cosine of the angle made by the rotation axis and Y-axis j/p
n_3 : Cosine of the angle made by the rotation axis and Z-axis k/p
θ : Angular displacement

Value p is obtained by the following:

$$p = \sqrt{i^2 + j^2 + k^2}$$

Conversion matrices for rotation on two-dimensional planes are shown below:

(1) Coordinate system conversion on the XY plane

$$M = \begin{pmatrix} \cos\theta & -\sin\theta & 0 \\ \sin\theta & \cos\theta & 0 \\ 0 & 0 & 1 \end{pmatrix}$$

(2) Coordinate system conversion on the YZ plane

$$M = \begin{pmatrix} 1 & 0 & 0 \\ 0 & \cos\theta & -\sin\theta \\ 0 & \sin\theta & \cos\theta \end{pmatrix}$$

(3) Coordinate system conversion on the ZX plane

$$M = \begin{pmatrix} \cos\theta & 0 & \sin\theta \\ 0 & 1 & 0 \\ -\sin\theta & 0 & \cos\theta \end{pmatrix}$$

- Three basic axes and their parallel axes

3-dimensional coordinate system conversion can be applied to a desired combination of three axes selected out of the basic three axes (X, Y, Z) and their parallel axes. The 3-dimensional coordinate system subjected to 3-dimensional coordinate system conversion is determined by axis addresses specified in the G68 block. If Xp, Yp, or Zp is not specified, X, Y, or Z of the basic three axes is assumed. However, if the basic three axes are not specified in parameter 1022, alarm PS0048, "BASIC 3 AXIS NOT FOUND" occurs.
In a single G68 block, both a basic axis and a parallel axis cannot be specified.
If this is attempted, alarm PS0047, "ILLEGAL AXIS SELECT" occurs.
(Example)

When U-axis, V-axis, and W-axis are parallel to the X-axis, Y-axis, and Z-axis respectively

```
G68  X_ I_ J_ K_ R_ ;          XYZ coordinate system
G68  U_V_ Z_ I_ J_ K_ R_ ;  UVZ coordinate system
G68  W_ I_ J_ K_ R_ ;          XYW coordinate system
```

- Specifying the second conversion

3-dimensional coordinate system conversion can be executed twice. The center of rotation of the second conversion must be specified with the axis addresses specified for the first conversion. If the axis addresses of the second conversion are different from the axis addresses of the first conversion, the different axis addresses are ignored. An attempt to execute 3-dimensional coordinate system conversion three or more times causes alarm PS5043, "TOO MANY G68 NESTING".

- Angular displacement R

A positive angular displacement R indicates a clockwise rotation along the axis of rotation. Specify angular displacement R in 0.001 degrees within the range of -360000 to 360000.
To specify angular displacement R in 0.00001 degrees (one hundred-thousandth), set bit 0 (FRD) of parameter No. 11630 to 1. In this case, angular displacement R is specified within the range of -36000000 to 36000000.

- G codes that can be specified

The following G codes can be specified in the 3-dimensional coordinate system conversion mode:

G00 Positioning
G01 Linear interpolation
G02 Circular interpolation (clockwise)
G03 Circular interpolation (counterclockwise)
G04 Dwell
G10 Data setting
G17 Plane selection (XY)
G18 Plane selection (ZX)
G19 Plane selection (YZ)
G28 Reference position return
G29 Movement from the reference position

G30 Second, third, or fourth reference position return
G31 Skip function
G53 Selecting the machine coordinate system
G65 Custom macro call
G66 Custom macro modal call
G67 Custom macro modal call cancel
G40 Canceling tool radius / tool nose radius compensation
G41 Tool radius / tool nose radius compensation to the left
G42 Tool radius / tool nose radius compensation to the right
G73, G74, G76, G80 to G89 Canned cycle for drilling

M

G43 Increasing tool length compensation
G44 Decreasing tool length compensation
G45 Increasing the tool offset
G46 Decreasing the tool offset
G47 Doubling the tool offset
G48 Halving the tool offset
G49(49.1) Canceling tool length compensation
G50.1 Canceling programmable mirror image
G51.1 Programmable mirror image
G90 Absolute programming
G91 Incremental programming
G94 Feed per minute
G95 Feed per revolution
G98 Canned cycle (return to the initial level)
G99 Canned cycle (return to the level of point R)

T

G90 Absolute programming (when G code system B or C is used.)
G91 Incremental programming (when G code system B or C is used.)
G94 Feed per minute (when G code system B or C is used.)
G95 Feed per revolution(when G code system B or C is used.)
G98 Canned cycle (return to the initial level) (when G code system B or C is used.)
G99 Canned cycle (return to the level of point R) (when G code system B or C is used.)

- Rapid traverse in a canned cycle for drilling

By using bit 1 (D3R) of parameter No. 11221, rapid traverse in the drilling direction in a canned cycle for drilling in the tilted working plane indexing mode, 3-dimensional coordinate conversion mode, or workpiece setting error compensation mode can be selected and the rapid traverse mode for shift operation in fine boring (G76) or back boring (G87) for the M system can also be selected. When bit 1 (D3R) of parameter No. 11221 is set to 0, the cutting feed mode is selected; when it is set to 1, the rapid traverse mode is selected. For positioning to the initial level at the start of a canned cycle for drilling, however, the rapid traverse mode is used, regardless of whether bit 1 (D3R) of parameter No. 11221 is set to 0 or 1. Rapid traverse override can be applied in the rapid traverse mode, and cutting feed override can be applied in the cutting feed mode. bit 1 (D3R) of parameter No. 11221 can also be used to change the mode of rapid traverse in the rigid tapping direction.

For acceleration/deceleration after interpolation

When bit 1 (D3R) of parameter No. 11221 is set to 1 (for the rapid traverse mode), rapid traverse in the drilling direction in a canned cycle for drilling in the tilted working plane indexing mode, 3-dimensional coordinate conversion mode, or workpiece setting error compensation mode is accelerated/decelerated along each axis independently according to the time constants specified in parameters Nos. 1620 and 1621. So, when performing positioning of linear interpolation type with bit 1 (D3R) of parameter No. 11221 set to 1, make the following settings:

Bit 1 of (LRP) of parameter No. 1401=1	Selects positioning of linear interpolation type.
Bit 4 of (PRT) of parameter No. 1603=1	Selects acceleration/deceleration of constant time type.
Parameter No. 1620	Set the same value for all.
Parameter No. 1621	Set the same value for all.

For acceleration/deceleration before interpolation

In a canned cycle for drilling, linear rapid traverse in the drilling direction is performed at all times.

Feedrate

When bit 1 (D3R) of parameter No. 11221 is set to 1 (for the rapid traverse mode), the rapid traverse rate in the drilling direction in a canned cycle for drilling in the tilted working plane indexing mode, 3-dimensional coordinate conversion mode, or workpiece setting error compensation mode is clamped to a parameter value indicated below if the rapid traverse override is 100%. In the rapid traverse override F0 mode, however, the rapid traverse rate is clamped to the value of parameter No. 1421 or a parameter value listed in Table 14.2 (a), whichever smaller.

Table 14.2 (a) Feedrate in the rapid traverse mode

Parameter No. 5412=0	Parameter No. 1420
Parameter No. 5412≠0	Value of parameter No. 1420 or value of parameter 5412, whichever smaller

However, when the external deceleration function is enabled, the feedrate is clamped to a parameter value listed in Table 14.2 (a) that is overridden or an external deceleration speed, whichever smaller.

NOTE
1 Set a rapid traverse rate for each axis in parameter No. 1420 and a tangential rapid traverse rate in parameter No. 5412.
2 If rapid traverse of nonlinear interpolation type is performed, the feedrate is clamped to the value of parameter No. 1420, regardless of the setting of parameter No. 5412.
3 When the cutting feed mode is selected (when bit 1 (D3R) of parameter No. 11221 is set to 0), set a tangential rapid traverse rate in parameter No. 5412.

- Compensation functions

M

If tool length compensation, tool radius / tool nose radius compensation, or tool offset is specified with 3-dimensional coordinate system conversion, compensation is performed first, followed by 3-dimensional coordinate system conversion.

T

If tool radius / tool nose radius compensation is specified with 3-dimensional coordinate system conversion, compensation is performed first, followed by 3-dimensional coordinate system conversion.

- Relationship between 3-dimensional and two-dimensional coordinate system conversion

3-dimensional and two-dimensional coordinate system conversion use identical G codes (G68 and G69). A G code specified with I, J, and K is processed as the command for 3-dimensional coordinate system conversion. A G code not specified with I, J, and K is processed as the command for two-dimensional coordinate system conversion.

- Custom macro system variables

Coordinates on the workpiece coordinate system are assigned to system variables #5041 to #5048 (current position on each axis).

- Reset

If a reset occurs during 3-dimensional coordinate system conversion mode, the mode is canceled and the continuous-state G code is changed to G69.
Bit 2 (D3R) of parameter No. 5400 determines whether just the G69.1 code is used to cancel the 3-dimensional coordinate system conversion mode (G68.1). When this setting is selected, a CNC reset by a reset operation or by an input signal from the PMC will not cancel the 3-dimensional coordinate system conversion mode.

- Absolute position display

The absolute coordinates based on the program or workpiece coordinate system can be displayed in the 3-dimensional coordinate system conversion mode. Specify a desired coordinate system in bit 6 (DAK) of parameter No. 3106.

- 3-dimensional rigid tapping

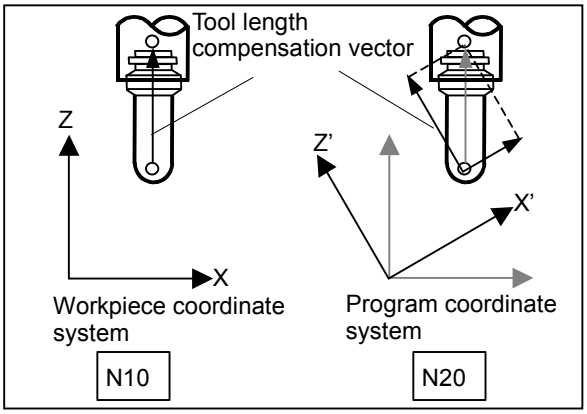
By specifying the rigid tapping command in 3-dimensional coordinate system conversion mode, tapping can be executed in the direction of the angle programmed by the 3-dimensional coordinate system conversion command.
In 3-dimensional coordinate system conversion mode, "Position Error Z", displayed on the spindle adjustment screen, is taken from the longitudinal tapping axis after 3-dimensional conversion.
Positioning in 3-dimensional coordinate system conversion mode must be linear interpolation positioning (bit 1 (LRP) of parameter No.1401 is set to 1).

- 3-dimensional coordinate system conversion during tool length compensation

By setting bit 3 (TLC) of parameter No. 11221 to 1, 3-dimensional coordinate system conversion can be used during tool length compensation. In this case, the tool length compensation vector is converted to the program coordinate system subject to 3-dimensional coordinate system conversion. This makes machining possible while keeping the tool direction to the Z-axis direction of the workpiece coordinate system.
Tool length compensation and 3-dimensional coordinate system conversion must be nested inside each other.

(Example)

```
N10 G43 H1 X_ Y_ Z_ ;
    :
N20 G68 X0 Y0 Z0 I0. J1. K0. R-30. ;
    :
G69 ;
    :
G49 X_ Y_ Z_ ;
```



Limitation

- Manual intervention

3-dimensional coordinate system conversion does not affect the degree of manual intervention or manual handle interrupt.

- Positioning in the machine coordinate system

3-dimensional coordinate system conversion does not affect positioning in the machine coordinate system (e.g. specified with G28, G30, or G53).

- Specifying rapid traverse

Specify linear rapid traverse when 3-dimensional coordinate system conversion is executed. (Set bit 1 (LRP) of parameter No.1401 to 1.)

- Block with G68 or G69

In a block with G68 or G69, other G codes must not be specified. G68 must be specified with I, J, and K.

- Position display and compensation

To display the absolute position when 3-dimensional coordinate system conversion is executed, set bits 4 to 7 (DRL, DRC, DAL, and DAC) of parameters No. 3104 to 0.

- Mirror image

M

Programmable mirror image can be specified, but external mirror image (mirror image by the mirror image signal or setting) cannot be specified. 3-dimensional coordinate system conversion is carried out after the programmable mirror image function is executed.

T

External mirror image (mirror image by the mirror image signal or setting) cannot be specified.

- 3-dimensional coordinate system conversion and other continuous-state commands

M

Canned cycles, G41, G42, or G51.1 must be nested between G68 and G69.
(Example)

```
G68 X100. Y100. Z100. I0. J0. K1. R45. ;
  :
G41 D01 ;
  :
G40 ;
  :
G69 ;
```

T

Canned cycles, G41 or G42 must be nested between G68.1 and G69.1.
(Example)

```
G68.1 X100. Y100. Z100. I0. J0. K1. R45. ;
  :
G41 X_ Z_ I_ K_ ;
  :
G40 ;
  :
G69.1 ;
```

T

- Relationship between 3-dimensional coordinate system conversion and tool offset

When using a tool offset command, nest the tool offset command within the 3-dimensional coordinate system conversion mode.
(Example)

```
G68.1 X100. Y100. Z100. I0. J0. K1. R45. ;
  :
T0101;
  :
T0100;
  :
G69.1 ;
```

- PMC axis control

In the 3-dimensional coordinate system conversion mode, PMC axis control cannot be performed for the three axes related to the conversion (alarm).

- Manual operation

When manual feeding is performed during a 3-dimensional coordinate system conversion, the tangent velocity in the coordinate system after conversion (program coordinate system) equals the lowest feedrate of those on the selected axes.

- Workpiece coordinate system

Avoid changing the workpiece coordinate system in the 3-dimensional coordinate system conversion mode.

- Manual reference position return

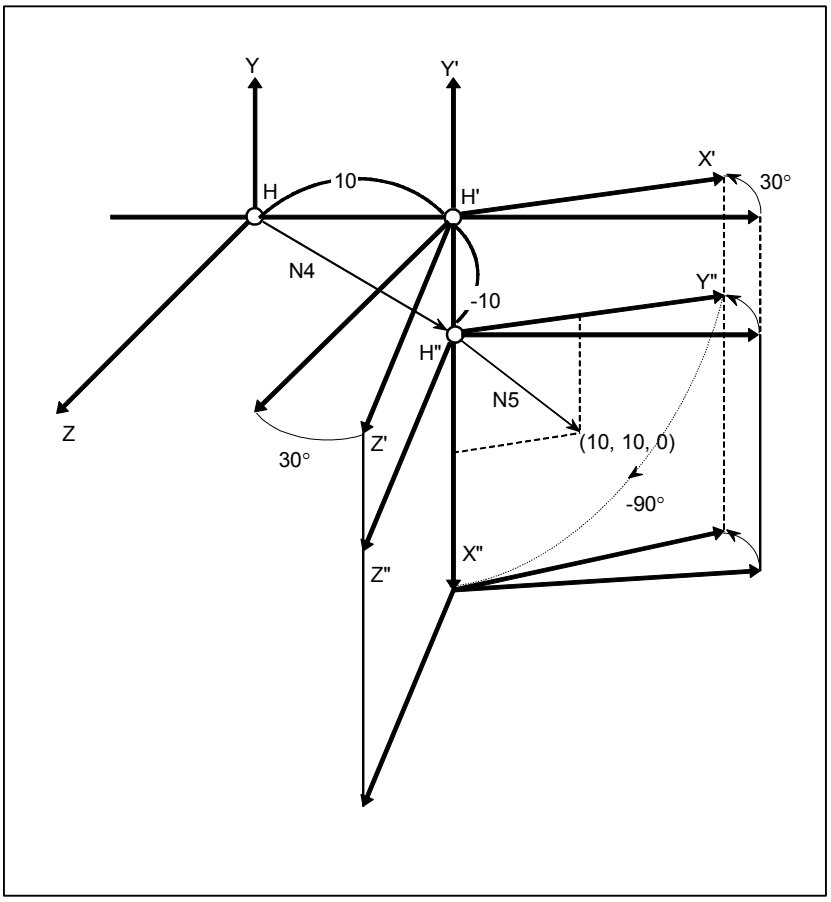
If manual reference position return is performed in the 3-dimensional coordinate system conversion mode, alarm PS5324 occurs. If you want to perform manual reference position return, cancel the 3-dimensional coordinate system conversion mode in advance.

- Cs contour axis

When specifying the Cs contour axis and rapid traverse at the same time in the 3-dimensional coordinate system conversion mode, make a reference position return on the Cs contour axis in advance. If a reference position return is made in the first rapid traverse after the Cs contour axis is selected (bit 1 (NRF) of parameter No.3700 is set to 0), avoid specifying the reference position return command in the 3-dimensional coordinate system conversion mode.

Example

N1 G90 X0 Y0 Z0 ;	Carries out positioning to zero point H.
N2 G68 X10. Y0 Z0 I0 J1 K0 R30. ;	Forms new coordinate system X'Y'Z'.
N3 G68 X0 Y-10. Z0 I0 J0 K1 R-90. ;	Forms other coordinate system X"Y"Z". The origin agrees with (0, -10, 0) in coordinate system X'Y'Z.
N4 G90 X0 Y0 Z0 ;	Carries out positioning to zero point H" on coordinate system X"Y"Z".
N5 X10. Y10. Z0 ;	Carries out positioning to (10, 10, 0) on coordinate system X"Y"Z".



15 COMPENSATION FUNCTION

Chapter 15, "COMPENSATION FUNCTION", consists of the following sections:

15.1 TOOL LENGTH COMPENSATION (G43, G44, G49).....295
15.2 SCALING (G50, G51).....302
15.3 PROGRAMMABLE MIRROR IMAGE (G50.1, G51.1).....309
15.4 NORMAL DIRECTION CONTROL (G40.1,G41.1,G42.1).....311
15.5 WORKPIECE SETTING ERROR COMPENSATION 315
15.6 TOOL OFFSET FOR MILLING AND TURNING FUNCTION (G43.7).....361
15.7 TOOL OFFSET CONVERSION FUNCTION (G44.1) 367

15.1 TOOL LENGTH COMPENSATION (G43, G44, G49)

This function can be used by setting the difference between the tool length assumed during programming and the actual tool length of the tool used into the offset memory. It is possible to compensate the difference without changing the program.
Specify the direction of offset with G43 or G44. Select a tool length compensation value from the offset memory by entering the corresponding address and number (H code).

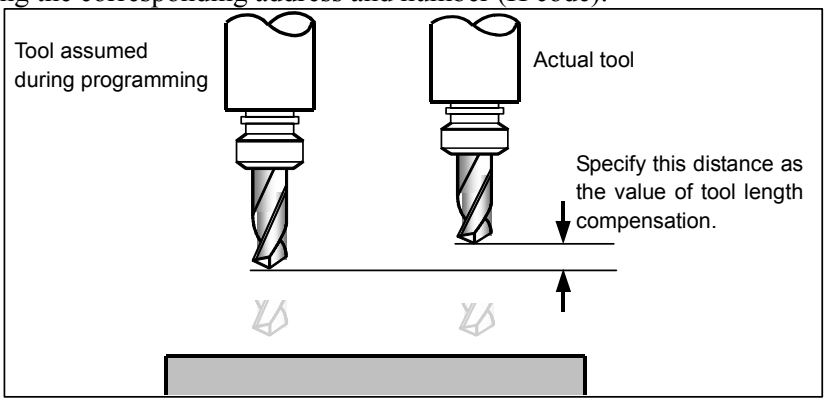


Fig. 15.1 (a) Tool length compensation

15.1.1 Overview

The following three methods of tool length compensation can be used, depending on the axis along which tool length compensation can be made.

- Tool length compensation A
  Compensates for the difference in tool length along the basic Z-axis.
- Tool length compensation B
  Compensates for the difference in tool length in the direction normal to a selected plane.
- Tool length compensation C
  Compensates for the difference in tool length along a specified axis.

Format

Type	Format	Description
Tool length compensation A	**G43 Z_ H_ ;** **G44 Z_ H_ ;**	G43 : Positive offset G44 : Negative offset G17 : XY plane selection G18 : ZX plane selection G19 : YZ plane selection α : Address of a specified axis H : Address for specifying the tool length compensation value X, Y, Z : Offset move command
Tool length compensation B	**G17 G43 Z_ H_ ;** **G17 G44 Z_ H_ ;** **G18 G43 Y_ H_ ;** **G18 G44 Y_ H_ ;** **G19 G43 X_ H_ ;** **G19 G44 X_ H_ ;**	
Tool length compensation C	**G43 α_H_ ;** **G44 α_H_ ;**	
Tool length compensation cancel	**G49 ; or H0 ;**	

Explanation

- Selection of tool length compensation

Select tool length compensation A, B, or C, by setting bits 0 (TLC) and 1 (TLB) of parameter No.5001 .

Parameter No.5001		Type
Bit 1 (TLB)	Bit 0 (TLC)	
0	0	Tool length compensation A
1	0	Tool length compensation B
0/1	1	Tool length compensation C

- Direction of the offset

When G43 is specified, the tool length compensation value (stored in offset memory) specified with the H code is added to the coordinates of the end position specified by a command in the program. When G44 is specified, the same value is subtracted from the coordinates of the end position. The resulting coordinates indicate the end position after compensation, regardless of whether the absolute or incremental mode is selected.

When the specification of an axis is omitted, a movement is made by the tool length compensation value.

G43 and G44 are modal G codes. They are valid until another G code belonging to the same group is used.

- Specification of the tool length compensation value

The tool length compensation value assigned to the number (offset number) specified in the H code is selected from offset memory and added to or subtracted from the moving command in the program.

Example

```
 :
H1 ;        The offset value of offset number 1 is selected.
 :
G43 Z_ ;    Offset is applied according to the offset value of offset number 1.
 :
H2 ;        Offset is applied according to the offset value of offset number 2.
 :
H0 ;        Offset is applied according to the offset value 0.
 :
H3 ;        Offset is applied according to the offset value of offset number 3.
 :
G49 ;       Offset is canceled.
 :
H4 ;        The offset value of offset number 4 is selected.
 :
```

A tool length compensation value is to be set in the offset memory corresponding to an offset number.

⚠ WARNING

When another offset number is specified, the tool length compensation value just changes to a new value. The new tool length compensation value is not added to the old tool length compensation value.

H1 : Tool length compensation value 20.0
H2 : Tool length compensation value 30.0
G90 G43 Z100.0 H1 ; Z will move to 120.0
G90 G43 Z100.0 H2 ; Z will move to 130.0

NOTE

The tool length compensation value corresponding to offset No. 0, that is, H0 always means 0. It is impossible to set any other tool length compensation value to H0.

- Performing tool length compensation along two or more axes

Tool length compensation B can be executed along two or more axes when the axes are specified in two or more blocks.

By setting bit 3 (TAL) of parameter No. 5001 to 1, tool length compensation C can also can be executed along two or more axes when the axes are specified in two or more blocks. If no axis is specified in the same block, the alarm PS0027, “NO AXES COMMANDED IN G43/G44” is issued. If two or more axes are specified in the same block, the alarm PS0336, “TOOL COMPENSATION COMMANDED MORE TWO AXES” is issued.

Example 1
When tool length compensation B is executed along the X-axis and Y-axis
G19 G43 H_ ; Offset in X axis
G18 G43 H_ ; Offset in Y axis

Example 2
When tool length compensation C is executed along the X-axis and Y-axis
G43 X_ H_ ; Offset in X axis
G43 Y_ H_ ; Offset in Y axis

Example 3
When an alarm is issued with tool length compensation C
G43 X_ Y_ H_ ; An alarm PS0336 occurs

- Tool length compensation cancel

To cancel tool length compensation, specify G49 or H0. After G49 or H0 is specified, the system immediately cancels the offset mode.

NOTE

1 If offset is executed along two or more axes, offset along all axes is canceled by specifying G49. If H0 is used to specify cancellation, offset along only the axis normal to a selected plane is canceled in the case of tool length compensation B, or offset along only the last axis specified by G43 or G44 is canceled in the case of tool length compensation C.

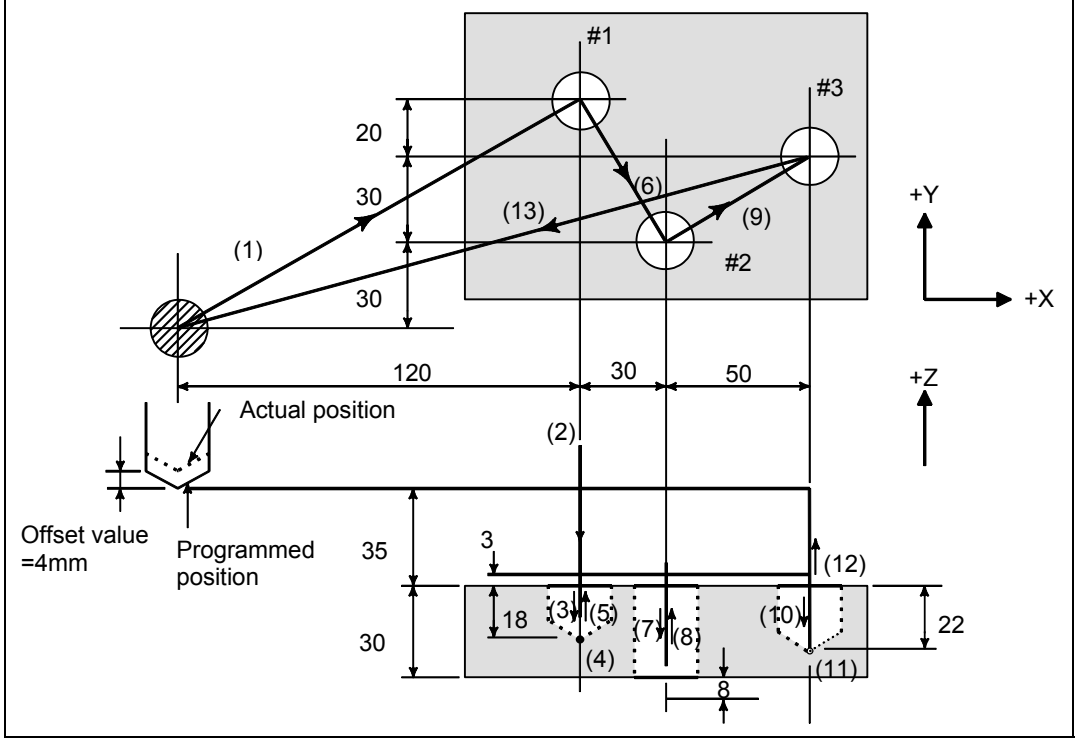
2 If offset is executed along three or more axes, and offset along all axes is canceled using G49, the alarm PS0015, “TOO MANY SIMULTANEOUS AXES” may be issued. By using H0 together, for example, cancel offset so that the number of simultaneously controlled axes (the number of axes along which movements are made simultaneously) does not exceed the allowable range of the system.

3 If tool length compensation and 3-dimensional coordinate system conversion are canceled by a reset with 3-dimensional coordinate system conversion performed during tool length compensation, the direction of tool length compensation vector cancellation becomes incorrect. By setting bit 6 (LVK) of parameter No. 5003 to 1 and setting bit 2 (D3R) of parameter No. 5400 to 1, ensure that the tool length compensation vector and 3-dimensional coordinate system conversion are not canceled by a reset.

```
Example)
 G43 H1 ;
 G68 X_ Y_ Z_ I_ J_ K_ R_ ;
   :
   :      } Ensure that tool length compensation and three-dimensional
   :      } coordinate conversion are not canceled by a reset in this range.
 G69 ;
 G49 ;
```

Example

Tool length compensation (in boring holes #1, #2, and #3)



Program
H1=-4.0 (Tool length compensation value)
N1 G91 G00 X120.0 Y80.0 ; .....(1)
N2 G43 Z-32.0 H1 ; .....(2)
N3 G01 Z-21.0 F1000 ; .....(3)
N4 G04 P2000 ; .....(4)
N5 G00 Z21.0 ; .....(5)
N6 X30.0 Y-50.0 ; .....(6)
N7 G01 Z-41.0 ; .....(7)
N8 G00 Z41.0 ; .....(8)
N9 X50.0 Y30.0 ; .....(9)
N10 G01 Z-25.0 ; .....(10)
N11 G04 P2000 ; .....(11)
N12 G00 Z57.0 H0 ; .....(12)
N13 X-200.0 Y-60.0 ; .....(13)
N14 M2 ;

Notes

- Command for setting a workpiece coordinate system in the tool length compensation mode

Executing a workpiece coordinate system setting G code command (G92 or, for G code system A in the lathe system, G50) presets a coordinate system in such a way that the specified position will be a pre-compensation position.
However, this G code cannot be used together with a block where tool length compensation vectors vary. For details, see “Notes” in Subsection 7.2.1, “Setting a Workpiece Coordinate System.”

15.1.2 G53, G28, G30, and G30.1 Commands in Tool Length Compensation Mode

This section describes the tool length compensation cancellation and restoration performed when G53, G28, G30, or G31 is specified in tool length compensation mode. Also described is the timing of tool length compensation.
As mentioned in "WARNINGS AND CAUTIONS RELATED TO PROGRAMMING" on page S-3 of this manual, it is recommended to cancel tool length compensation mode before executing the G53, G28, G30, and G30.1 commands.

Explanation

- Tool length compensation vector cancellation

When G53, G28, G30, or G30.1 is specified in tool length compensation mode, tool length compensation vectors are canceled as described below. However, the previously specified modal G code remains displayed; modal code display is not switched to G49.

(1) When G53 is specified

Command	Specified axis	Operation
G53 IP_	Tool length compensation axis	Canceled upon movement being performed
	Other than tool length compensation axis	Not canceled
G49 G53 IP_	Tool length compensation axis	Canceled upon movement being performed
	Other than tool length compensation axis	Canceled

(IP_ : Dimension word)

> ⚠ **CAUTION**
> If tool length compensation is applied along multiple axes, the offset vector along the axis specified by G53 is canceled.

(2) When G28, G30, or G30.1 is specified

Command	Specified axis	Operation
G28 IP_	Tool length compensation axis	Not canceled at an intermediate point. Canceled at the reference position.
	Other than tool length compensation axis	Not canceled at an intermediate point. Canceled at the reference position.
G49 G28 IP_	Tool length compensation axis	Canceled when a movement is made to an intermediate point.
	Other than tool length compensation axis	Canceled when a movement is made to an intermediate point.

(IP_ : Dimension word)

> ⚠ **CAUTION**
> If tool length compensation is applied along multiple axes, the offset vector along the axis on which a reference position return operation has been performed is canceled.

- Tool length compensation vector restoration

Tool length compensation vectors, canceled by specifying G53, G28, G30, or G30.1 in tool length compensation mode, are restored as described below.

Type	Bit 6 (EVO) of parameter No. 5001	Restoration condition
A/B	0	The H command or G43 (G44) is specified.
	1	Restored by the next buffered block.
C		The H command or G43 (G44)IP_ is specified.

(IP_ : Dimension word)

⚠ CAUTION

1 If a tool length compensation vector is restored only with H_, G43, or G44 when tool length compensation is applied along multiple axes, the tool length compensation vector along only the axis normal to a selected plane is restored in the case of tool length compensation B, or the tool length compensation vector along only the last axis for which tool length compensation is specified is restored in the case of tool length compensation C. The tool length compensation vector along any other axes is not restored.

2 In the block in which the tool length compensation vector is restored, do not execute commands other than positioning with G00 or G01, G04, and a single block with EOB.

15.2 SCALING (G50, G51)

Overview

A programmed figure can be magnified or reduced (scaling).
Two types of scaling are available, one in which the same magnification rate is applied to each axis and the other in which different magnification rates are applied to different axes.
The magnification rate can be specified in the program.
Unless specified in the program, the magnification rate specified in the parameter is applied.

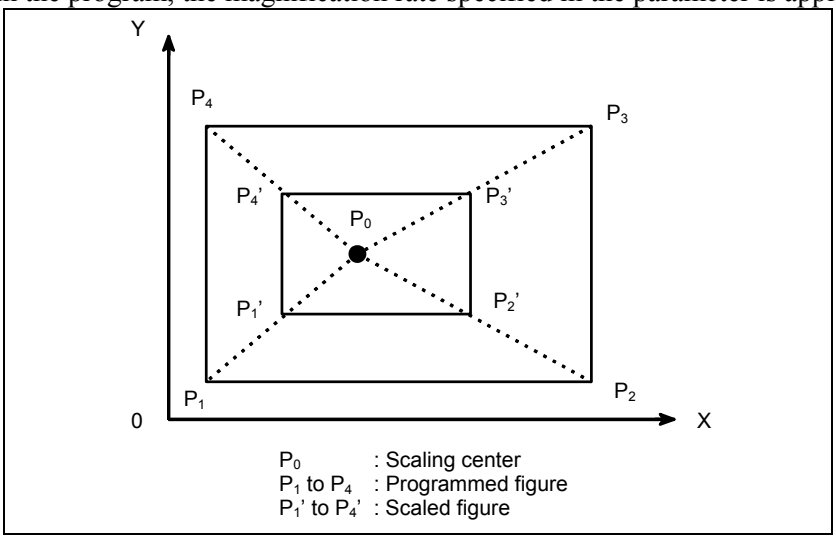


Fig. 15.2 (a) Scaling

Format

Scaling up or down along all axes at the same rate of magnification
(When bit 6 (XSC) of parameter No. 5400 = 0)

Format	Meaning of command
G51 IP_P_ ; Scaling start ⎫ ⎫ Scaling is effective. ⎭ ⎭ (Scaling mode) **G50 ;** Scaling cancel	**IP_** : Absolute command for center coordinate value of scaling **P_** : Scaling magnification

Scaling up or down along each axes at a different rate of magnification (mirror image)
(When bit 6 (XSC) of parameter No. 5400 = 1)

Format	Meaning of command
G51 IP_I_J_K_; Scaling start ⎫ ⎫ Scaling is effective. ⎭ ⎭ (Scaling mode) **G50 ;** Scaling cancel	**IP_** : Absolute command for center coordinate value of scaling **I_J_K_** : Scaling magnification for basic 3 axes (X, Y, and Z axes) respectively

T

NOTE
This function is available when the G-code system B or C is set.

⚠ CAUTION
1 Specify G51 in a separate block.
2 After the figure is enlarged or reduced, specify G50 to cancel the scaling mode.

NOTE
1 Entering electronic calculator decimal point input mode (bit 0 (DPI) of parameter No. 3401 = 1) does not cause the units of the magnification rates P, I, J, and K to change.
2 Setting the least input increment equal to 10 times the least command increment (bit 7 (IPR) of parameter No. 1004 = 1) does not cause the units of the magnification rates P, I, J, and K to change.
3 An attempt to specify 0 as a magnification rate causes alarm PS0142, “ILLEGAL SCALE RATE” to occur in a G51 block.

Explanation

- Axis for which scaling is to be enabled

For the axis for which scaling is to be enabled, set bit 0 (SCL) of parameter No. 5401 to 1.

- Minimum unit of scaling magnification

Least input increment of scaling magnification is: 0.001 or 0.00001.
It is 0.00001 (one hundred thousandth) if bit 7 (SCR) of parameter No. 5400 is 0 and 0.001 if it is 1.

- Scaling center

Even in incremental command (G91) mode, the scaling center coordinates IP_ specified in the G51 block are assumed those of an absolute position.
If the scaling center coordinates are omitted, the position assumed when G51 is specified is assumed the scaling center.

⚠ CAUTION
With the move command subsequent to the G51 block, execute an absolute (G90 mode) position command.
If no absolute position command is executed after the G51 block, the position assumed when G51 is specified is assumed the scaling center; once an absolute position command is executed, the scaling center assumes the coordinates specified in the G51 block, after that block.

- Scaling along each axis at the same rate of magnification

Set bit 6 (XSC) of parameter No. 5400 to 0.
If the scaling magnification P is not specified, the magnification set in parameter No. 5411 is used.
Decimal point input is not accepted as the magnification P. If decimal point input is made, alarm PS0007, “ILLEGAL USE OF DECIMAL POINT” will occur.
A negative value cannot be specified as the magnification P. If a negative value is specified, alarm PS0006, “ILLEGAL USE OF MINUS SIGN” will occur.
The allowable magnification range is from 0.00001 to 9999.99999.

- Scaling of each axis, programmable mirror image (negative magnification)

Each axis can be scaled by different magnifications. Also when a negative magnification is specified, a mirror image is applied. The axis subject to the mirror image is the one that contains the scaling center.
Set bit 6 (XSC) of parameter No. 5400to 1 to validate each axis scaling (mirror image).
Using I, J, and K, specify the scaling magnifications for the basic 3 axes (X to Z axes). Use parameter No. 1022 to specify which axes to use as the basic 3 axes. For those of the X to Z axes for which I, J, and K are not specified and for axes other than the basic 3 axes, the magnification set with parameter No. 5421 is used.
A value other than 0 must be set to parameter No. 5421.
Decimal point programming can not be used to specify the rate of magnification (I, J, K).
Magnification can be set within the range of ±0.00001 to ±9999.99999.

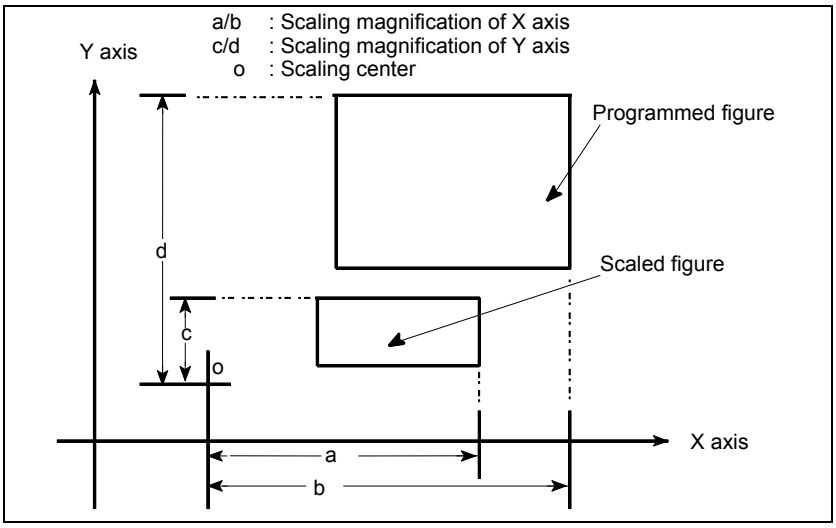


Fig. 15.2 (b) Scaling of each axis

⚠ CAUTION

Specifying the following commands at the same time causes them to be executed in the order indicated below:

<1> Programmable mirror image (G51.1)
<2> Scaling (G51) (including a mirror image with a negative magnification)
<3> Mirror image due to the external switch of the CNC or the settings of the CNC

In this case, the programmable mirror image is effective to the scaling center and magnification as well.

To specify G51.1 and G51 at the same time, specify them in this order; to cancel them, specify them in the reverse order.

- Scaling of circular interpolation

Even if different magnifications are applied to each axis in circular interpolation, the tool will not trace an ellipse.

```
G90 G00 X0.0 Y100.0 Z0.0;
G51 X0.0 Y0.0 Z0.0 I2000 J1000;
```

(A magnification of 2 is applied to the X-component and a magnification of 1 is applied to the Y-component.)

```
G02 X100.0 Y0.0 I0 J–100.0 F500;
```

Above commands are equivalent to the following command:

```
G90 G00 X0.0 Y100.0 Z0.0;
G02 X200.0 Y0.0 I0 J–100.0 F500;
```

(Because the end point is not on an arc, spiral interpolation is assumed.)

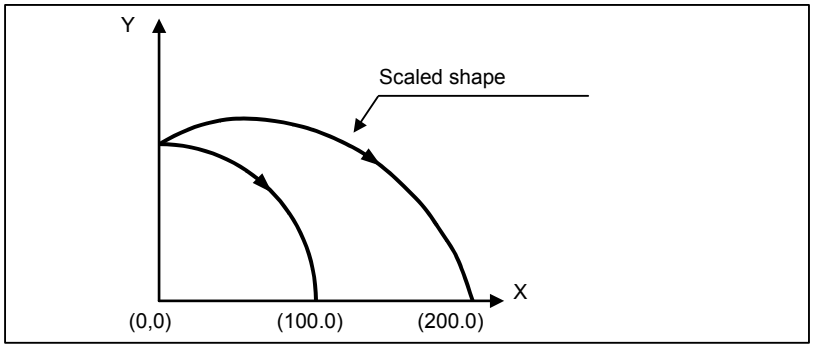


Fig. 15.2 (c) Scaling for circular interpolation

Even for an R-specified arc, scaling is applied to each of I, J, and K after the radius value (R) is converted into a vector in the center direction of each axis.
If, therefore, the above G02 block contains the following R-specified arc, the operation will be same as that in which I and J are specified.

G02 X100.0 Y0.0 R100.0 F500 ;

- Scaling and coordinate system rotation

If both scaling and coordinate system rotation are specified at the same time, scaling is performed first, followed by coordinate system rotation. In this case, scaling is effective to the rotation center as well.
To specify both of them, specify scaling first and then coordinate system rotation. To cancel them, specify them in the reverse order.

Example
Main program

```
O1
G90 G00 X20.0 Y10.0 ;
M98 P1000 ;
G51 X20.0 Y10.0 I3000 J2000 ;  (×3 in the X direction and ×2 in the Y direction)
M98 P1000 ;
G17 G68 X35.0 Y20.0 R30. ;
M98 P1000 ;
G69 ;
G50 ;
M30 ;
```

Subprogram

```
O1000 ;
G01 X20.0 Y10.0 F500 ;
G01 X50.0 ;
G01 Y30.0 ;
G01 X20.0 ;
G01 Y10.0 ;
M99 ;
```

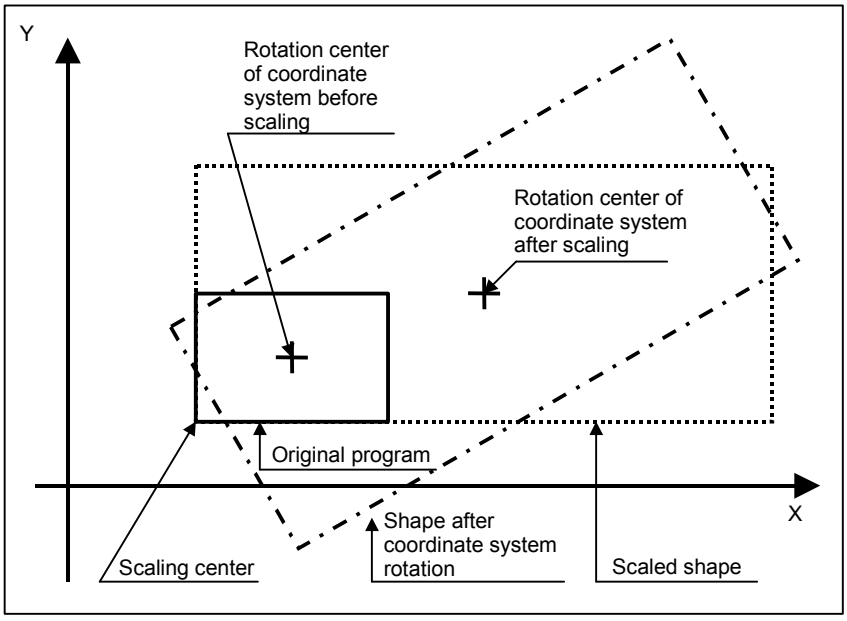


Fig. 15.2 (d) Scaling and coordinate system rotation

- Scaling and optional chamfering/corner R

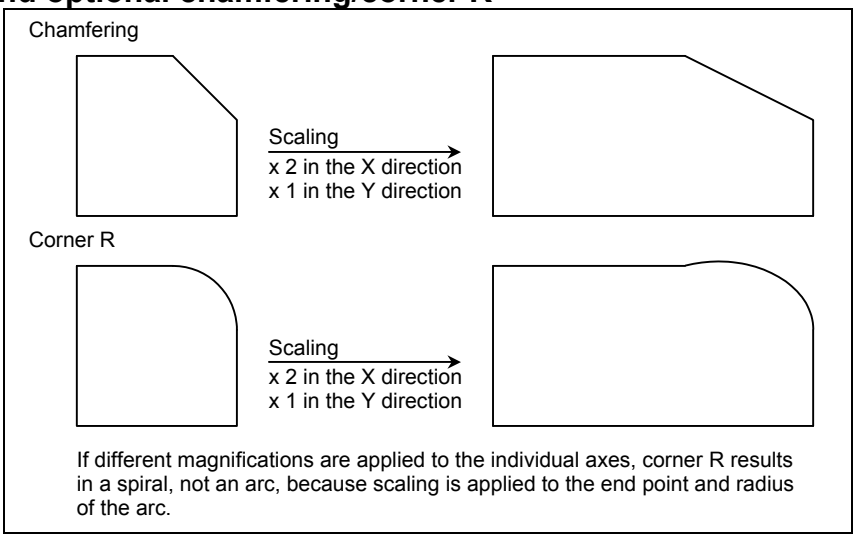


Fig. 15.2 (e) Scaling and optional chamfering/corner R

Limitation

- Tool compensation

This scaling is not applicable to tool radius · tool nose radius compensation values, tool length compensation values, and tool offset values (Fig. 15.2 (f)).

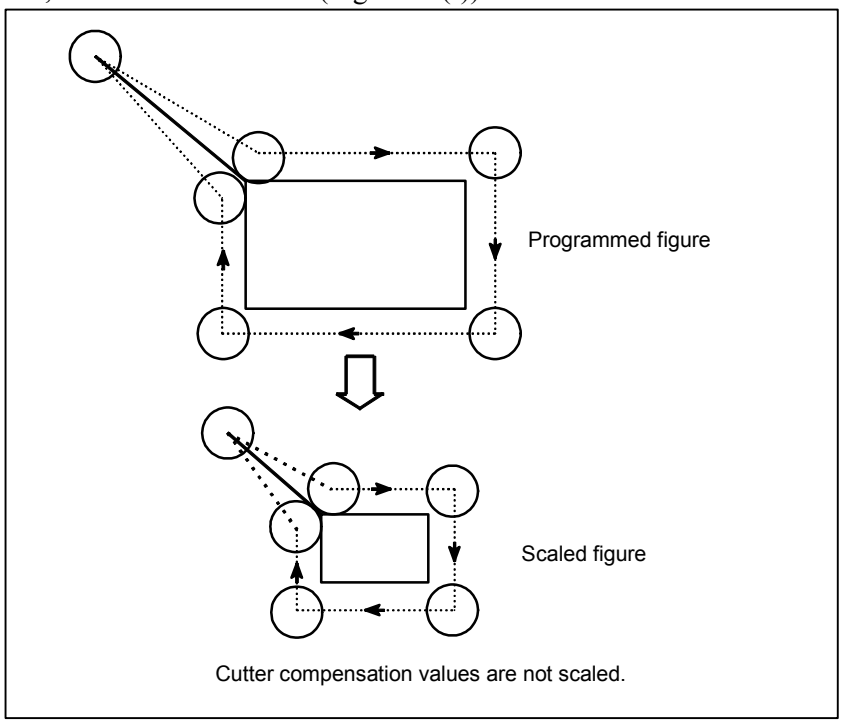


Fig. 15.2 (f) Scaling during cutter compensation

- Invalid scaling

M

Scaling is not applied to the travel distance during canned cycle shown below.

- Cut-in value Q and retraction value d of peck drilling cycle (G83, G73).
- Fine boring cycle (G76)
- Shift value Q of X and Y axes in back boring cycle (G87).

In manual operation, the travel distance cannot be increased or decreased using the scaling.

T

This function is available to G code systems B and C only; it is not available to G code system A.
During scaling, the following functions cannot be used. If any of them is specified, alarm PS0300, "ILLEGAL COMMAND IN SCALING" will occur.

- Finishing cycle (G70, G72)
- Outer surface rough cutting cycle (G71, G73)
- End rough cutting cycle (G72, G74)
- Closed loop cutting cycle (G73, G75)
- End cutting off cycle (G74, G76)
- Outer/inner surface cutting off cycle (G75, G77)
- Multiple threading cycle (G76, G78)
- Traverse grinding cycle (for grinding machines) (G71, G72)
- Traverse direct constant-size grinding cycle (for grinding machines) (G72, G73)
- Oscillation grinding cycle (for grinding machines) (G73, G74)
- Oscillation direct constant-size grinding cycle (for grinding machines) (G74, G75)
- Face drilling cycle (G83, G83)
- Face tapping cycle (G84, G84)
- Face boring cycle (G85, G85)
- Side drilling cycle (G87, G87)
- Side tapping cycle (G88, G88)
- Side boring cycle (G89, G89)
- Outer/inner surface turning cycle (G77, G20)
- Threading cycle (G78, G21)
- End turning cycle (G79, G24) (The G code in system B is given first, followed by that of system C.)

In manual operation, the travel distance cannot be increased or decreased using the scaling.
If, during scaling, tool position compensation with a coordinate system shift (bit 2 (LWT) of parameter No. 5002 = 1 or bit 4 (LGT) of parameter No. 5002 = 0) is specified, alarm PS0509, "TOOL OFFSET COMMAND IS NOT AVAILABLE", is issued. This is also true if bit 6 (EVO) of parameter No. 5001 is set to 1 and the tool position compensation value is changed.

⚠ CAUTION

1 If a parameter setting value is employed as a scaling magnification without specifying P, the setting value at G51 command time is employed as the scaling magnification, and a change of this value, if any, is not effective.
2 Before specifying the G code for reference position return (G27, G28, G29, G30, etc.) or coordinate system setting (G52 to G59, G92, etc.), cancel the scaling mode.
3 If scaling results are rounded by counting fractions of 5 and over as a unit and disregarding the rest, the move amount may become zero. In this case, the block is regarded as a no movement block, and therefore, it may affect the tool movement by cutter compensation. See the description of cutter compensation.
4 Refrain from scaling on a rotation axis for which the rollover function is enabled. Otherwise, the tool may rotate in a short-cut manner, possibly resulting in unexpected movement.

NOTE
1 The position display represents the coordinate value after scaling.
2 When a mirror image was applied to one axis of the specified plane, the following results:
(1) Circular command ..... Direction of rotation is reversed.
(2) Tool radius · tool nose radius compensation . Offset direction is reversed.
(3) Coordinate system rotation ..... Rotation angle is reversed.

Example

Sample program of a scaling in each axis

```
O1;
G51 X20.0 Y10.0 I750 J250; (× 0.75 in the X direction, × 0.25 in the Y direction)
G00 G90 X60.0 Y50.0;
G01 X120.0 F100;
G01 Y90;
G01 X60;
G01 Y50;
G50;
M30;
```

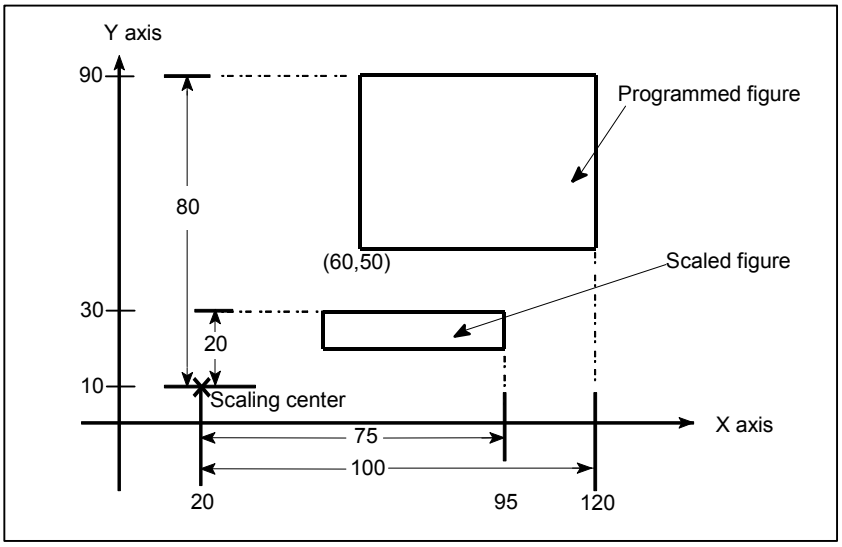
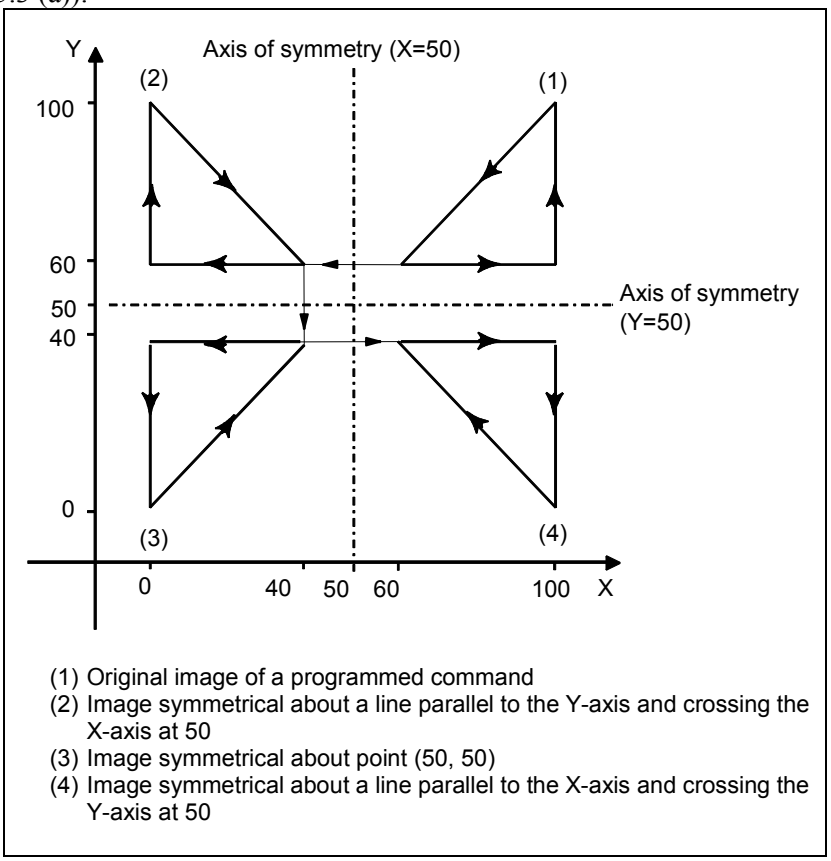


Fig. 15.2 (g) Program example of scaling in each axis

15.3 PROGRAMMABLE MIRROR IMAGE (G50.1, G51.1)

A mirror image of a programmed command can be produced with respect to a programmed axis of symmetry (Fig. 15.3 (a)).



(1) Original image of a programmed command
(2) Image symmetrical about a line parallel to the Y-axis and crossing the X-axis at 50
(3) Image symmetrical about point (50, 50)
(4) Image symmetrical about a line parallel to the X-axis and crossing the Y-axis at 50

Fig. 15.3 (a) Programmable mirror image

Format

G51.1 IP_ ; Setting a programmable image

:
:
: A mirror image of a command specified in these blocks is produced with respect to the axis of symmetry specified by G51.1 IP_;.
:

G50.1 IP_ ; Canceling a programmable mirror image

IP_ : Point (position) and axis of symmetry for producing a mirror image when specified with G51.1.
Axis of symmetry for producing a mirror image when specified with G50.1. Point of symmetry is not specified.

Explanation

- Mirror image by setting

If the programmable mirror image function is specified when the command for producing a mirror image is also selected by a CNC external switch or CNC setting (see Section, “MIRROR IMAGE”), the programmable mirror image function is executed first.

- Mirror image on a single axis in a specified plane

Applying a mirror image to one of the axes on a specified plane changes the following commands as follows :

Command	Explanation
Circular command	G02 and G03 are interchanged.
Tool radius · tool nose radius compensation	G41 and G42 are interchanged.
Coordinate system rotation	CW and CCW (directions of rotation) are interchanged.

Limitation

- Scaling and coordinate system rotation

Processing proceeds from program mirror image to scaling and coordinate system rotation in the stated order. The commands should be specified in this order, and, for cancellation, in the reverse order. Do not specify G50.1 or G51.1 during scaling or coordinate system rotation mode.

- Commands related to reference position return and coordinate system

In programmable mirror image mode, G codes related to reference position return (G27, G28, G29, G30, etc.) and those for changing the coordinate system (G52 to G59, G92, etc.) must not be specified. If any of these G codes is necessary, specify it only after canceling the programmable mirror image mode.

T

If, during the programmable mirror image mode, tool position compensation with a coordinate system shift (bit 2 (LWT) of parameter No. 5002 = 1 or bit 4 (LGT) of parameter No. 5002 = 0) is specified, alarm PS0509, "TOOL OFFSET COMMAND IS NOT AVAILABLE", is issued.
This is also true if bit 6 (EVO) of parameter No. 5001 is set to 1 and the tool position compensation value is changed.

15.4 NORMAL DIRECTION CONTROL (G40.1,G41.1,G42.1)

Overview

When a tool with a rotation axis (C-axis) is moved in the XY plane during cutting, the normal direction control function can control the tool so that the C-axis is always perpendicular to the tool path (Fig. 15.4 (a)).

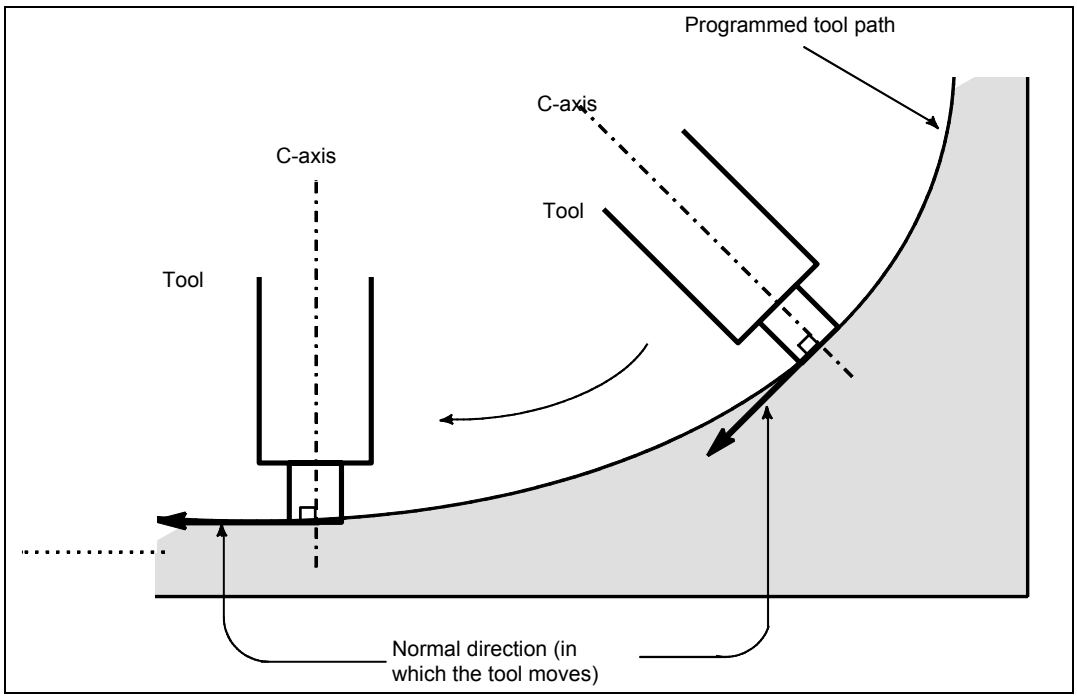


Fig. 15.4 (a) Sample Movement of the tool

Format

G41.1 ; Normal direction control, left

G42.1 ; Normal direction control, right

G40.1 ; Cancel normal direction control

The normal direction control, left (G41.1) command is used when the workpiece is on the right side of the tool as viewed while you are looking into the tool's way.
Once either G41.1 or G42.1 is issued, normal direction control is enabled (normal direction control mode).
Issuing G40.1 cancels the normal direction control mode.

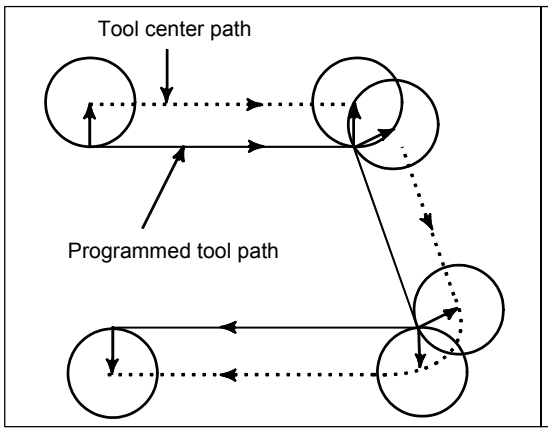


Fig. 15.4 (b) Normal direction control, left (G41.1)

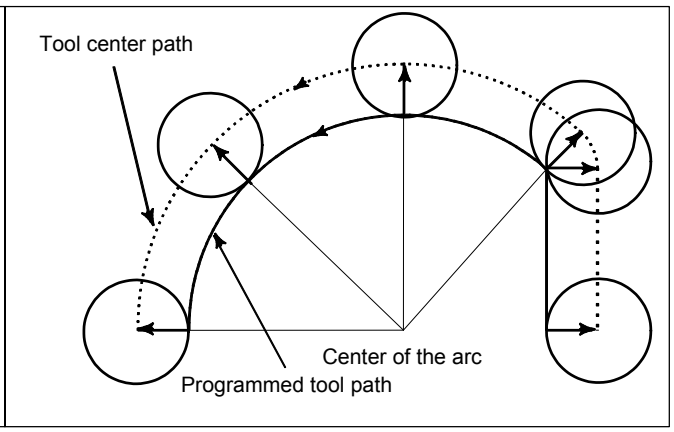


Fig. 15.4 (c) Normal direction control, right (G42.1)

Explanation

- Angle of the C axis

When viewed from the center of rotation around the C-axis, the angular displacement about the C-axis is determined as shown in Fig. 15.4 (d). The positive side of the X-axis is assumed to be 0, the positive side of the Y-axis is 90°, the negative side of the X-axis is 180°, and the negative side of the Y-axis is 270°.

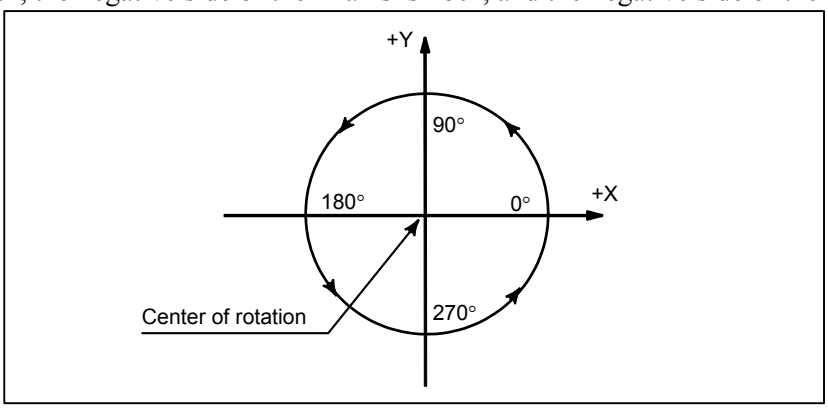


Fig. 15.4 (d) Angle of the C axis

- Normal direction control of the C axis

When the cancel mode is switched to the normal direction control mode, the C-axis becomes perpendicular to the tool path at the beginning of the block containing G41.1 or G42.1.

In the interface between blocks in the normal direction control mode, a command to move the tool is automatically inserted so that the C-axis becomes perpendicular to the tool path at the beginning of each block. The tool is first oriented so that the C-axis becomes perpendicular to the tool path specified by the move command, then it is moved along the X- and Y axes.

In the cutter compensation mode, the tool is oriented so that the C-axis becomes perpendicular to the tool path created after compensation.

In single-block operation, the tool is not stopped between a command for rotation of the tool and a command for movement along the X- and Y-axes. A single-block stop always occurs after the tool is moved along the X- and Y-axes.

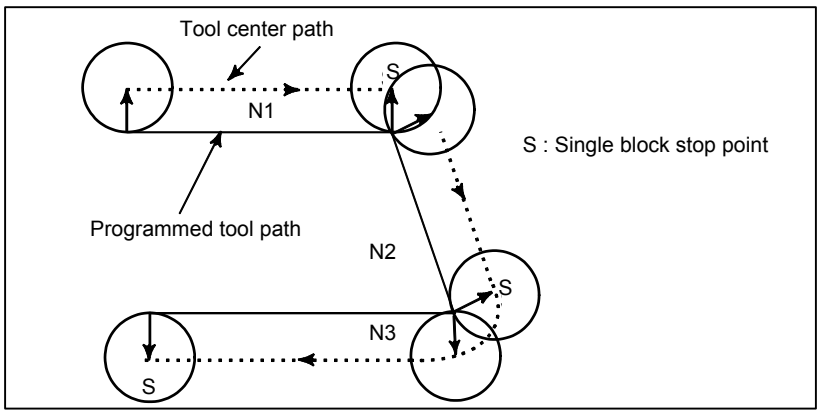


Fig. 15.4 (e) Point at which a single-block stop occurs in the normal direction control mode

Before circular interpolation is started, the C-axis is rotated so that the C-axis becomes normal to the arc at the start point. During circular interpolation, the tool is controlled so that the C-axis is always perpendicular to the tool path determined by circular interpolation.

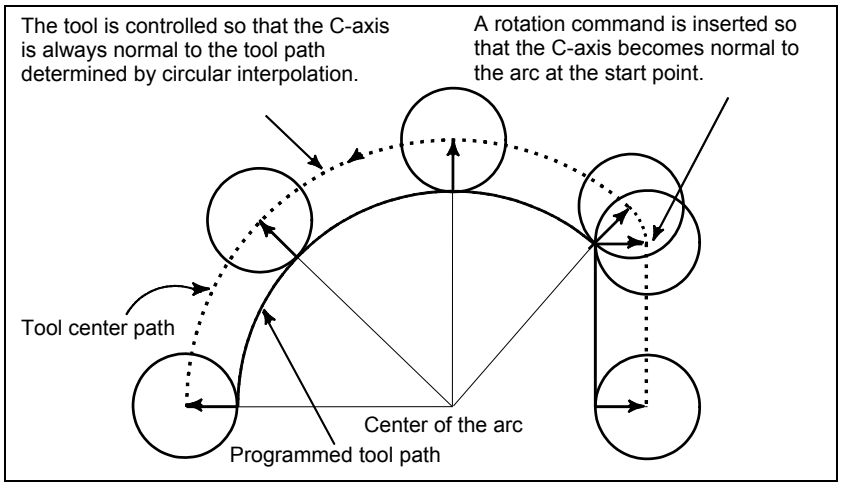


Fig. 15.4 (f) Normal direction control of the circular interpolation

NOTE
During normal direction control, the C axis always rotates through an angle less than 180 deg. I.e., it rotates in whichever direction provides the shorter route.

- C axis feedrate

Movement of the tool inserted at the beginning of each block is executed at the feedrate set in parameter 5481. If dry run mode is on at that time, the dry run feedrate is applied. If the tool is to be moved along the X-and Y-axes in rapid traverse (G00) mode, the rapid traverse feedrate is applied.
The feedrate of the C axis during circular interpolation is defined by the following formula.

$$F \times \frac{\text{Amount of movement of the C axis (deg)}}{\text{Length of arc (mm or inch)}} \quad \text{(deg/min)}$$

F : Feedrate (mm/min or inch/min) specified by the corresponding block of the arc
Amount of movement of the C axis : The difference in angles at the beginning and the end of the block.

NOTE
If the feedrate of the C axis exceeds the maximum cutting speed of the C axis specified to parameter No. 1430, the feedrate of each of the other axes is clamped to keep the feedrate of the C axis below the maximum cutting speed of the C axis.

- Normal direction control axis

A C-axis to which normal-direction control is applied can be assigned to any axis with parameter No. 5480.

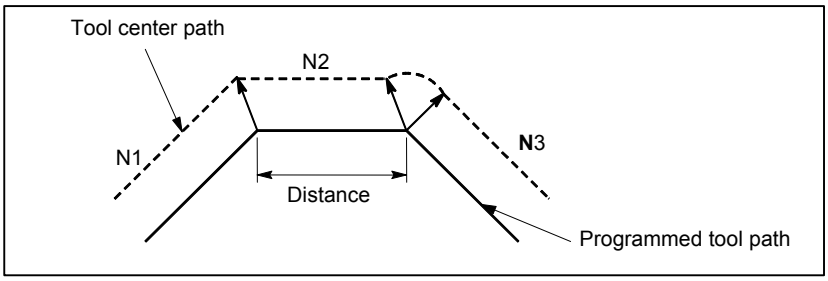
- Angle for which figure insertion is ignored

When the rotation angle to be inserted, calculated by normal-direction control, is smaller than the value set with parameter No. 5482, the corresponding rotation block is not inserted for the axis to which normal-direction control is applied. This ignored rotation angle is added to the next rotation angle to be inserted, the total angle being subject to the same check at the next block.
If an angle of 360 degrees or more is specified, the corresponding rotation block is not inserted.
If an angle of 180 degrees or more is specified in a block other than that for circular interpolation with a C-axis rotation angle of 180 degrees or more, the corresponding rotation block is not inserted.

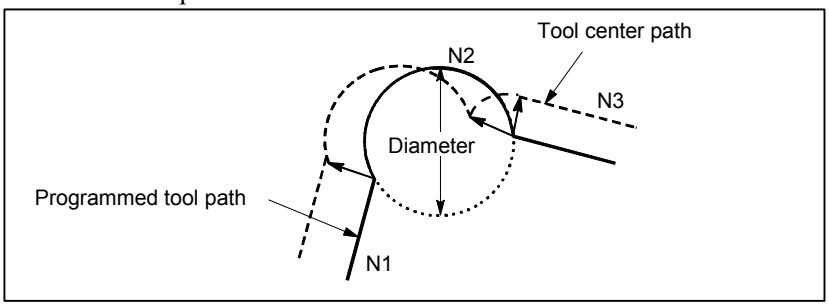
- Movement for which arc insertion is ignored

Specify the maximum distance for which machining is performed with the same normal direction as that of the preceding block.

- Linear movement
  When distance N2, shown below, is smaller than the set value, machining for block N2 is performed using the same direction as that for block N1.



- Circular movement
  When the diameter of block N2, shown below, is smaller than the set value, machining for block N2 is performed using the same normal direction as that for block N1. And control as compensation along the circular arc is not performed.



NOTE
1 Do not specify any command to the C axis during normal direction control. Any command specified at this time is ignored.
2 Before processing starts, it is necessary to correlate the workpiece coordinate of the C axis with the actual position of the C axis on the machine using the coordinate system setting (G92) or the like.
3 The helical cutting option is required to use this function. Helical cutting cannot be specified in the normal direction control mode.
4 Normal direction control cannot be performed by the G53 move command.
5 The C-axis must be a rotation axis.

T

NOTE
In the normal direction control mode, the following commands cannot be issued. An attempt to issue any of them results in alarm PS1471 being raised.
- Plane selection command (G17, G18, G19)
- Automatic reference position return(G28)
- 2nd/3rd/4th reference position return(G30)
- Move from the reference position (G29)

15.5 WORKPIECE SETTING ERROR COMPENSATION

When a workpiece is placed on the machine, the workpiece is not always placed at an ideal position.
With this function, a displaced workpiece can be machined according to the program.
This function can compensate for a workpiece setting error on a rotation axis used with a function involving rotation axis operation such as a function for tool center point control, function for 3-dimensional cutter compensation, and a tilted working plane command. So, a workpiece can be machined according to the program even during tool center point control, 3-dimensional cutter compensation, and tilted working plane command.

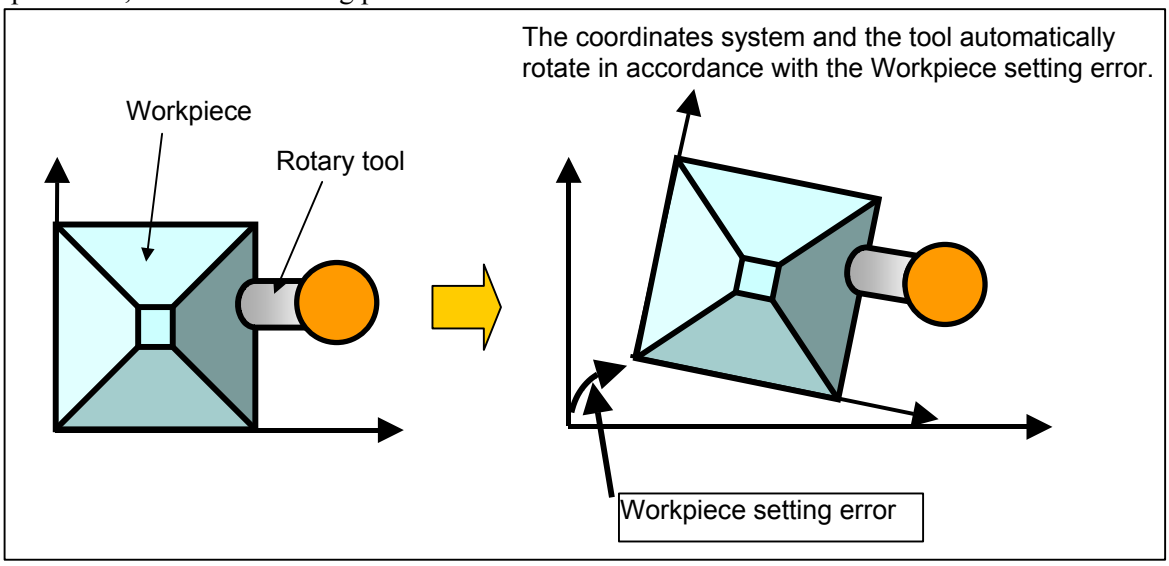


Fig. 15.5 (a) Outline of workpiece setting error

Format

```
G54.4 Pn ;      Start workpiece setting error compensation
  :
  :             Workpiece setting error compensation mode
G54.4 P0 ;      Cancel workpiece setting error compensation

  Pn  : Workpiece setting error specification code
        n  : 1 to 7
```

A start block and cancellation block for workpiece setting error compensation suppress buffering.

Explanation

- Workpiece setting error

A workpiece setting error is defined by the following eight variables:

- X direction error Δx
- Y direction error Δy
- Z direction error Δz
- Rotation direction error Δa (rotation error on the X-axis in degrees)
- Rotation direction error Δb (rotation error on the Y-axis in degrees)
- Rotation direction error Δc (rotation error on the Z-axis in degrees)
- Table rotation axis position 1
- Table rotation axis position 2

[About Δx, Δy, and Δz]

Δx, Δy, and Δz represent the coordinate values of the origin of the workpiece coordinate system (X'Y'Z' in the Fig. 15.5 (b), which is hereinafter referred to as the "workpiece setting coordinate system") based on a displaced workpiece, as viewed from the original workpiece coordinate system (XYZ in the Fig. 15.5 (b)).

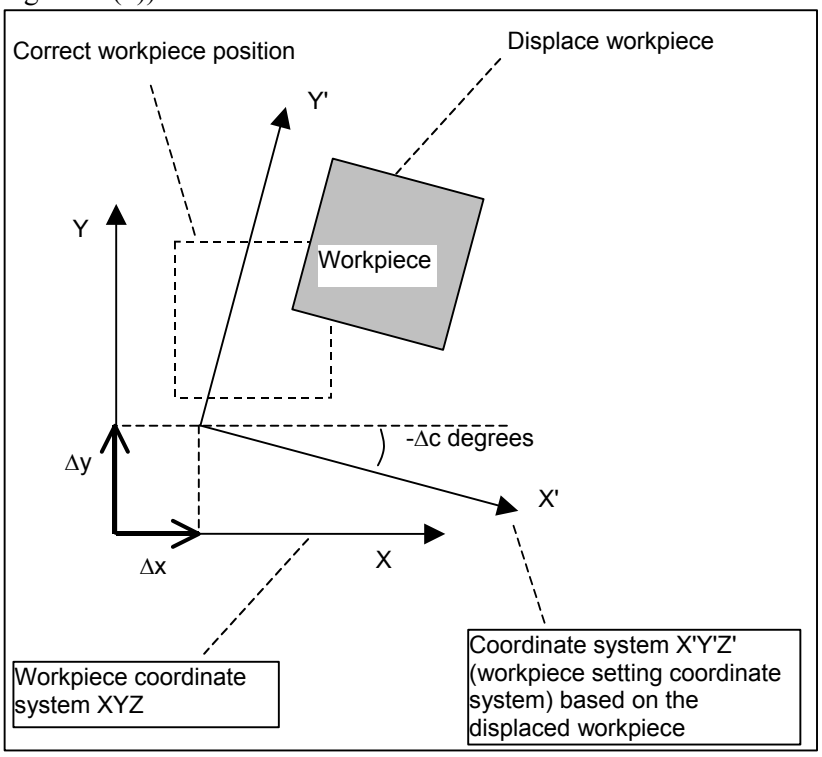


Fig. 15.5 (b) Errors in X, Y, and Z directions

[About Δa, Δb, and Δc]

Δa, Δb, and Δc are defined as described below.

The workpiece coordinate system obtained by rotating a workpiece coordinate system about the X-axis by angle Δa, about the Y-axis by angle Δb, and about the Z-axis by angle Δc then shifting that workpiece coordinate system by (Δx, Δy, Δz) from the origin of that workpiece coordinate system is supposed to match the "workpiece setting coordinate system".

(Note: In determination of positive/negative rotation, clockwise rotation is defined to be positive rotation.)

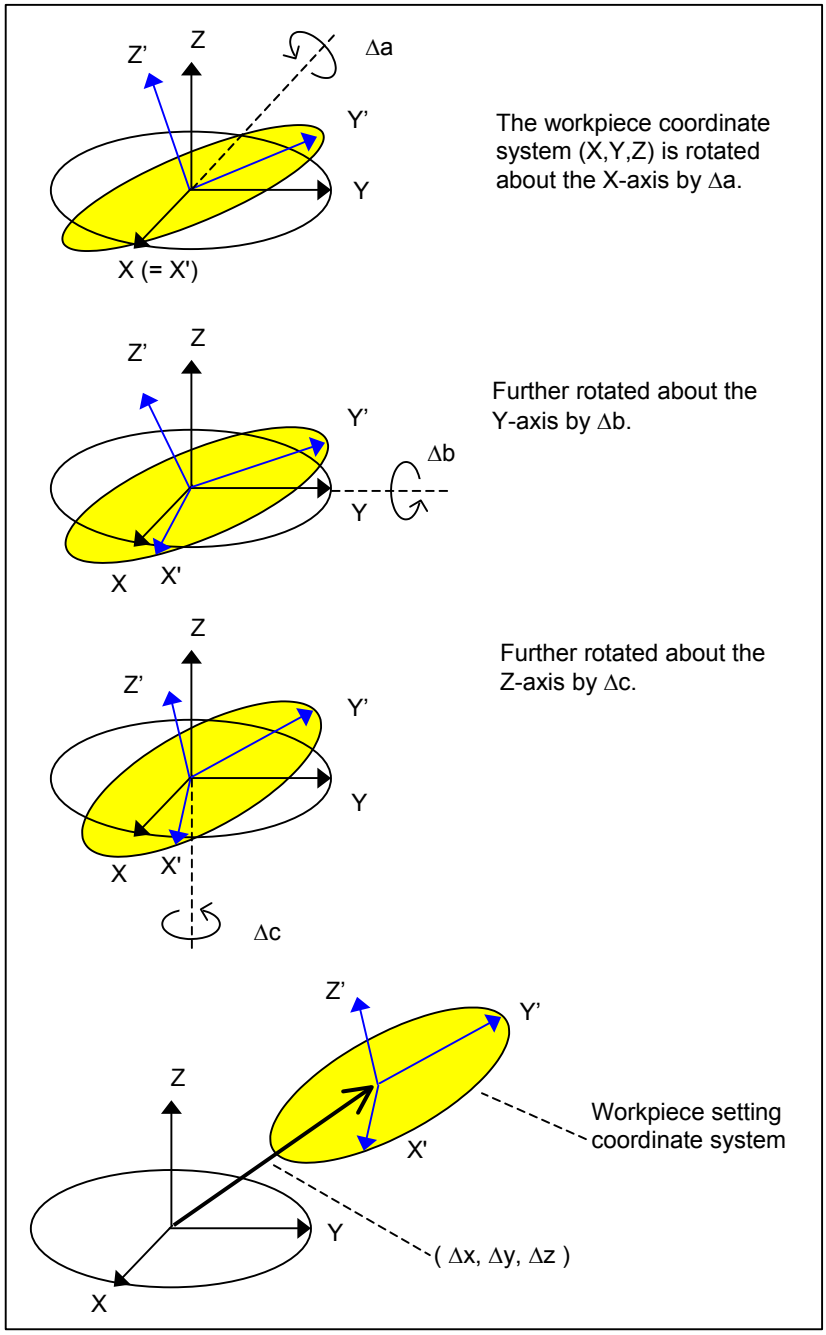


Fig. 15.5 (c) Errors in rotation directions

The example of Fig. 15.5 (b) assumes that Δa = Δb = 0 and Δc represents a nonzero value.

[About table rotation axis position 1 and table rotation axis position 2]

The table rotation axis position means the machine coordinate on the table rotation axis of a 5-axis machine with a table rotation axis (machine of table rotation type or composite type) when a workpiece setting error (displacement from the correct workpiece position to the actual workpiece position) is measured.

When two table rotation axes are used, set table rotation axis position 1 for the master rotation axis, and set table rotation axis position 2 for the slave rotation axis. (As for master rotation axis and slave rotation axis, refer to the Table 15.5 (k), “"Rotation axis closer to the tool” and "rotation axis closer to the workpiece"” described later.)

When only one table rotation axis is used, set table rotation axis position 1 for the axis.

When no table rotation axis is used, or the machine used is not a 5-axis machine, table rotation axis position 1 and table rotation axis position 2 need not be set.
No setting can be made for a hypothetical axis.

In the descriptions above, X, Y, and Z represent the three basic axes, X, Y, and Z, specified by parameter No. 1022.
If the specification of any of the three basic axes, X, Y, and Z, is missing, alarm PS0436, “ILLEGAL PARAMETER IN WSC” is issued.

- Setting of a workpiece setting error on the workpiece setting error setting screen

A workpiece setting error is set on the workpiece setting error setting screen.
For explanations about the setting method, see Chapter 12, "SETTING AND DISPLAYING DATA," of Part III.

- Workpiece setting error of No. 00 (COMMON)

Each of the values set in No. 00 (COMMON) is added to the corresponding value of each of No. 01 through No. 07.
Note that rotation direction errors for No.00(COMMON) are not available.
Example:

Suppose that workpiece setting errors are set as follows:

Table 15.5 (a)

No. 00 (COMMON)	No. 01	No. 02
X 10.000	x 0.500	x 0.800
Y 0.000	y 0.000	y 0.000
Z 0.000	z 0.000	z 0.000
	a 1.500	a 1.800
	b 0.000	b 0.000
	c 0.000	c 0.000

When the workpiece setting error of No. 01 is selected, workpiece setting error compensation is performed based on the following:
$\Delta x = 10.000+0.500 = 10.500$
$\Delta a = 1.500$

When the workpiece setting error of No. 02 is selected, workpiece setting error compensation is performed based on the following:
$\Delta x = 10.000+0.800 = 10.800$
$\Delta a = 1.800$

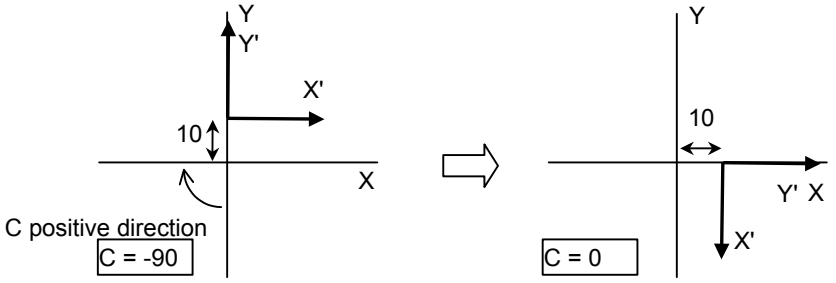
If the setting of table rotation axis position 1/table rotation axis position 2 differs between the workpiece setting error of No. 00 and each workpiece setting error when a 5-axis machine (machine of table rotation type or composite type) with table rotation axes is used, no simple additions are made. Instead, before additions are made, conversion is made to values based on the machine coordinate on a table rotation axis being 0.
Example:

Suppose that workpiece setting errors are set as follows on a 5-axis machine with table rotation axis C about the Z-axis.

Table 15.5 (b)

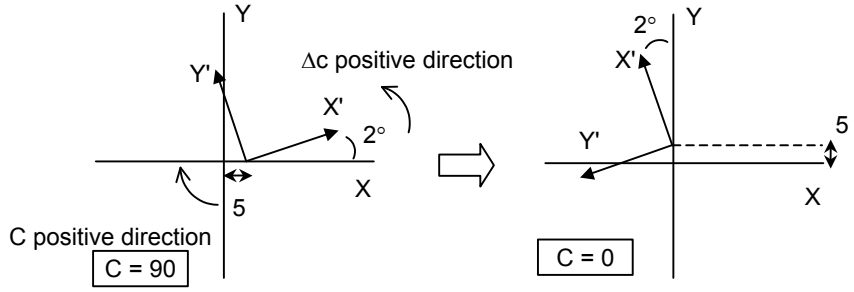
No. 00 (COMMON)	No. 01
X 0.000	x 5.000
y 10.000	y 0.000
z 0.000	z 0.000
	a 0.000
	b 0.000
	c 2.000
C -90.000	C 90.000

First, the error values of No. 00 are converted to those based on C = 0.000.



Δy = 10.000 Δx = 10.000

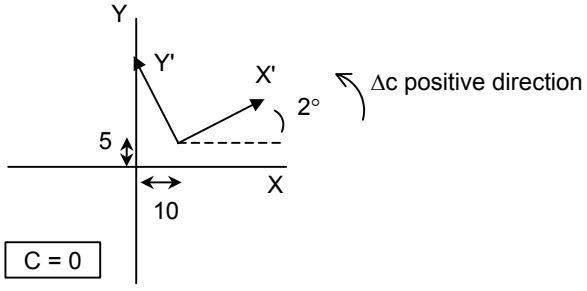
Next, the error values of No. 01 are converted to those based on C = 0.000.



Δx = 5.000, Δc = 2.000 Δy = 5.000, Δc = 2.000

When the workpiece setting error of No. 01 is selected, workpiece setting error compensation is performed as follows based on the converted error values based on C = 0.000:

Δx = 10.000+0.000 = 10.000
Δy = 0.000+5.000 = 5.000
Δc = 2.000



Δx = 10.000, Δy = 5.000, Δc = 2.000

- Least input increment of data and valid data range

[About errors Δx, Δy, and Δz]

The unit of data is the unit of input, and follows the least input increment for the reference axis specified by parameter No. 1031.

Table 15.5 (c) For metric input

Unit system of reference axis	IS-A	IS-B	IS-C	IS-D	IS-E
Least input increment (mm)	0.01	0.001	0.0001	0.00001	0.000001
Maximum settable value (mm)	±999,999.99	±999,999.999	±99,999.9999	±9,999.99999	±999.999999

Table 15.5 (d) For inch input

Unit system of reference axis	IS-A	IS-B	IS-C	IS-D	IS-E
Least input increment (inch)	0.001	0.0001	0.00001	0.000001	0.0000001
Maximum settable value (inch)	±99,999.999	±99,999.9999	±9,999.99999	±999.999999	±99.9999999

[About errors Δa, Δb, and Δc]
With parameter No. 11201, the number of decimal places of the least input increment can be specified.

Table 15.5 (e)

Parameter No. 11201	1	2	3	4
Least input increment (deg)	0.1	0.01	0.001	0.0001
Maximum settable value (deg)	±99,999,999.9	±9999,999.99	±999,999.999	±99,999.9999

Parameter No. 11201	5	6	7	8
Least input increment (deg)	0.00001	0.000001	0.0000001	0.00000001
Maximum settable value (deg)	±9,999.99999	±999.999999	±99.9999999	±9.99999999

Note, however, that a value from 1 to 8 can be specified in parameter No. 11201.
If a value not within the specifiable range is specified in parameter No. 11201, the least input increment of the reference axis is followed (Table 15.5 (f)).

Table 15.5 (f)

Unit system of reference axis	IS-A	IS-B	IS-C	IS-D	IS-E
Least input increment (deg)	0.01	0.001	0.0001	0.00001	0.000001
Maximum settable value (deg)	±999,999.99	±999,999.999	±99,999.9999	±9,999.99999	±999.999999

[About table rotation axis position 1 and table rotation axis position 2]
The unit of data follows the least input increment of each corresponding rotation axis.

Table 15.5 (g)

Unit system of rotation axis	IS-A	IS-B	IS-C	IS-D	IS-E
Least input increment (deg)	0.01	0.001	0.0001	0.00001	0.000001
Maximum settable value (deg)	±999,999.99	±999,999.999	±99,999.9999	±9,999.99999	±999.999999

- "Active workpiece setting error"

In the workpiece setting error compensation mode, "Active workpiece setting error" on the workpiece setting error setting screen displays the currently selected workpiece setting error number and workpiece setting error (sum of the data of the currently selected workpiece setting error number and the data of No. 00).

If the workpiece setting error compensation mode is not set, "MODE OFF" is indicated, and 0 is displayed in each data field.

If a 5-axis machine (machine of table rotation type or composite type) with table rotation axes is used, and a nonzero value is set for table rotation axis position 1/table rotation axis position 2, error values obtained by conversion based on the machine coordinates on the table rotation axes being 0 are displayed.

- Setting of workpiece setting errors with custom macro variables

By using custom macro variables #26000 to #26077, workpiece setting errors can be read and written.
(The custom macro option is required.)

The numbers of macro variables correspond to Errors as follows (Table 15.5 (h)) :

Table 15.5 (h)

	Error No.00 (COMMON)	Error No.01	Error No.02	Error No.03	Error No.04	Error No.05	Error No.06	Error No.07
X direction error Δx	#26000	#26010	#26020	#26030	#26040	#26050	#26060	#26070
Y direction error Δy	#26001	#26011	#26021	#26031	#26041	#26051	#26061	#26071
Z direction error Δz	#26002	#26012	#26022	#26032	#26042	#26052	#26062	#26072
Rotation direction error Δa	-	#26013	#26023	#26033	#26043	#26053	#26063	#26073
Rotation direction error Δb	-	#26014	#26024	#26034	#26044	#26054	#26064	#26074
Rotation direction error Δc	-	#26015	#26025	#26035	#26045	#26055	#26065	#26075
Table rotation axis position 1	#26006	#26016	#26026	#26036	#26046	#26056	#26066	#26076
Table rotation axis position 2	#26007	#26017	#26027	#26037	#26047	#26057	#26067	#26077

- Workpiece setting error compensation mode

By specifying G54.4 Pn (n: 1 to 7), the workpiece setting error compensation mode is set.
With Pn, select a workpiece setting error from No. 01 to No. 07.
In the workpiece setting error compensation mode, the program is executed in a "workpiece setting coordinate system" defined by shifting the workpiece coordinate system.
G54.4 is a modal G code that belongs to group 33 with the M series or group 26 with the T series.
By specifying G54.4 P0, the workpiece setting error compensation mode is canceled.

In a block for starting workpiece setting error compensation, the machine is not moved, but the absolute coordinates are changed by the error. So, the next absolute command makes a movement to a specified position in the workpiece setting coordinate system. This means that an absolute command is needed after a block for starting workpiece setting error compensation.

NOTE

1 Be sure to specify Pn in a block specifying G54.4. If P is not specified, or a number (n) not within the specifiable range is specified after P, alarm PS0437, “ILLEGAL COMMAND IN WSC” is issued.
2 Specify G54.4 singly. If another G code or axis command is specified together, alarm PS0437 is issued.
3 Workpiece setting error compensation is valid for movement in automatic operation.
4 Any workpiece setting error between starting blocks remains valid until the workpiece setting error compensation mode is canceled.
5 Workpiece setting error compensation may not be specified doubly. If G54.4 Pn (n 0) is specified in the workpiece setting error compensation mode, alarm PS0437 is issued.
6 No absolute coordinate is changed in a block for starting workpiece setting error compensation unless the two angles stated below exceed the value specified in parameter No. 11204.
   (1) Angle made by the current absolute coordinate point and a singular point posture
   (2) Angle made by a point sifted by a workpiece setting error from the absolute coordinate point and a singular point posture
7 If all the errors are 0, issuing the G54.4 command cannot put the workpiece setting error compensation mode in effect.

- Setting of workpiece setting errors with programmable data input (G10)

By inputting programmable data, workpiece setting errors can be set.

Format

G10 L23 P_ X_ Y_ Z_ A_ B_ C_ I_ J_;

P : Workpiece setting error number 0 to 7
X : X-direction error Δx
Y : Y-direction error Δy
Z : Z-direction error Δz
A : Rotational error Δa
B : Rotational error Δb
C : Rotational error Δc
I : Table rotation axis position 1
J : Table rotation axis position 2

NOTE
1 If an address other than P is omitted, the amount of workpiece setting error corresponding to the omitted address is assumed to be the amount before specification.
2 The specified value is used as the amount of workpiece setting error as is. Even when a value is specified in the G91 mode, it is used as the amount of workpiece setting error as is. The specified value is not added to the value before specification.
3 For workpiece setting error number 0 (COMMON), any rotational error cannot be set. If rotational error A, B, or C is specified for setting error number 0, alarm PS1144, "G10 FORMAT ERROR" is issued.
4 When there is only one table rotation axis, table rotation axis position 2 cannot be set. When there is no table rotation axis, both table rotation axis positions 1 and 2 cannot be set. If an attempt is made to set the position, alarm PS1144, “G10 FORMAT ERROR" is issued.
5 If an address other than P, X, Y, Z, A, B, C, I, or J is specified, alarm PS1144, “G10 FORMAT ERROR” is issued.
6 If a value other than 0 to 7 is specified for workpiece setting error number P, alarm PS0031, “ILLEGAL P COMMAND IN G10” is issued.
7 An NC address with no decimal point is assumed as follows depending on the setting of bit 0 (DPI) of parameter No. 3401:
0: Least input increment
1: Unit of mm or inches

Example)

G10 L23 P2 X5.0 Y0 Z0 A0 B0 C2.0 I90.0;

On a 5-axis machine with table rotation axis C about the Z-axis, the above command sets the following workpiece setting errors for workpiece setting error number 2:Δx = 5.0, Δy = 0.0, Δz = 0.0 Δa = 0.0, Δb = 0.0, Δc = 2.0, and table rotation axis position 1 = 90.0

NOTE
1 The set workpiece setting errors are used when the machine enters the workpiece setting error compensation mode next time after G54.4 is specified. When workpiece setting errors are set, they are not used as workpiece setting errors for workpiece setting error compensation being executed.
2 No workpiece setting error can be changed in simulation of dynamic graphic display or Manual Guide i.

- Settings on a 5-axis machine

With a 5-axis machine, the following parameters must be set:

Table 15.5 (i)

Parameter No.	Description
19680	Type of mechanical section
19681	Controlled axis number of the first rotation axis
19682	Axis direction of the first rotation axis
19683	Inclination angle when the first rotation axis is inclined
19684	Rotation direction of the first rotation axis
19685	Rotation angle when the first rotation axis is a hypothetical axis
19686	Controlled axis number of the second rotation axis
19687	Axis direction of the second rotation axis
19688	Inclination angle when the second rotation axis is inclined
19689	Rotation direction of the second rotation axis
19690	Rotation angle when the second rotation axis is a hypothetical axis
19696#0	Whether the first rotation axis is an ordinary rotation axis/hypothetical axis
19696#1	Whether the second rotation axis is an ordinary rotation axis/hypothetical axis
19697	Tool axis direction
19698	Reference angle RA
19699	Reference angle RB
19700	Rotary table position (X-axis of the basic three axes)
19701	Rotary table position (Y-axis of the basic three axes)
19702	Rotary table position (Z-axis of the basic three axes)
19703	Intersection offset vector between the first and second rotation axes of the table (X-axis of the basic three axes)
19704	Intersection offset vector between the first and second rotation axes of the table (Y-axis of the basic three axes)
19705	Intersection offset vector between the first and second rotation axes of the table (Z-axis of the basic three axes)

- When tool direction compensation is not performed

When bit 0 (RCM) of parameter No. 11200 is set to 0, tool direction compensation (compensation for a rotation axis) is not performed.
Set this parameter to 0 when the machine is not a 5-axis machine or when it is a 5-axis machine, but tool direction compensation (compensation for a rotation axis) is not required.
The tool tip position is correctly compensated, but the direction of the tool against the workpiece is shifted by the rotation direction errors (Δa, Δb, and Δc).
This mode has the advantage that the singular point described later does not need to be considered because the tool moves about the rotary axis as specified.

- When tool direction compensation is performed

With a 5-axis machine, tool direction compensation can be performed by setting bit 0 (RCM) of parameter No. 11200 to 1. This function performs compensation for the rotation axis position so that the direction of the tool against the workpiece becomes that specified in the program.
Set this parameter to 1 when drilling is performed, when the workpiece is machined with the side of a tool, or when the side of a tool will interfere with the workpiece if the tool direction is not compensated.

NOTE
1 Depending on the machine configuration, it may be physically impossible to orient the tool in the compensation direction. In such a case, alarm DS0030, “TOOL DIRC CMP IMPOSSIBLE” is issued.

NOTE
2 In a block for starting workpiece setting error compensation, the absolute coordinate on a rotation axis is changed considering the workpiece setting error. At this time, depending on the machine configuration, a rotation axis for orienting the tool in the tool direction as viewed from the workpiece setting coordinate system may be absent. In such a case, alarm PS0438, “ILLEGAL PARAMETER IN TOOL DIRC CMP” is issued.
3 When TCP(Tool center point control) or G53.1 of TWP(Tilted working plane command) is used during Workpiece setting error compensation, Tool direction compensation is needed (Bit 0 (RCM) of parameter No. 11200 is 1.) Otherwise, in the case of TCP, the alarm PS5421, “ILLEGAL COMMAND IN G43.4/G43.5”, in the case of G53.1 of TWP, the alarm PS5458, “ILLEGAL USE OF G53.1/G53.6” occurs.

- Acceleration/deceleration before look-ahead interpolation

Acceleration/deceleration before look-ahead interpolation is automatically enabled in the workpiece setting error compensation mode. For this reason, it is advisable to turn the AI contour control mode on (by specifying G05.1 Q1 or setting bit 0 (SHP) of parameter No. 1604 to 1), then specify workpiece setting error compensation. In this case, also set the following parameters related to acceleration/deceleration.

Table 15.5 (j)

Parameter No.	Description
1432	Maximum cutting feedrate in the acceleration/deceleration before interpolation mode
1660	Maximum allowable acceleration rate in acceleration/deceleration before interpolation
1671	Maximum allowable acceleration rate for rapid traverse in acceleration/deceleration before interpolation
1672	Acceleration/deceleration change time for rapid traverse in bell-shaped acceleration/deceleration before interpolation
1737	Allowable acceleration rate for each axis when the deceleration function based on acceleration/deceleration under AI contour control.
1769	Time constant for acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode
1772	Acceleration/deceleration change time in bell-shaped acceleration/deceleration before interpolation
1783	Allowable speed difference in speed determination based on a corner speed difference

NOTE
1 If any of the parameters above is set incorrectly, alarm PS0438, “ILLEGAL PARAMETER IN TOOL DIRC CMP” is issued.

NOTE

2 Set the following parameters for acceleration/deceleration before look-ahead interpolation since acceleration/deceleration before look-ahead interpolation is automatically enabled:
   (1) Bit 1 (LRP) of parameter No. 1401=1:
   Linear rapid traverse
   (2) Bit 5 (FRP) of parameter No. 19501=1:
   Use acceleration/deceleration before interpolation for rapid traverse.
   (3) Parameter No. 1671:
   Maximum allowable acceleration rate for rapid traverse in acceleration/deceleration before interpolation
   (4) Parameter No. 1660:
   Maximum allowable acceleration rate in acceleration/deceleration before interpolation.
   If the parameters above are not set, alarm PS0438 is issued.
   However, if the option of AI contour control or AI contour control II is not active and one of the following conditions is satisfied, setting (2), (3), (4) is not needed because acceleration/deceleration before look-ahead interpolation is not automatically enabled.
   - Both The first and second rotation axes are hypothetical axes
   (Bits 0 (AI1) and 0 (AI1) of parameter No. 19696 are 1)
   - The 3-axis machine
   (parameter No. 19680=0, Bit 1 (WS3) of parameter No.11768=1)

- "Rotation axis closer to the tool" and "rotation axis closer to the workpiece" on a 5-axis machine

When tool direction compensation is performed on a 5-axis machine, a singular point and singular point posture need to be considered. Here, a "rotation axis closer to the tool" and "rotation axis closer to the workpiece", which are used in a description of a singular point and singular point posture provided later, are explained.

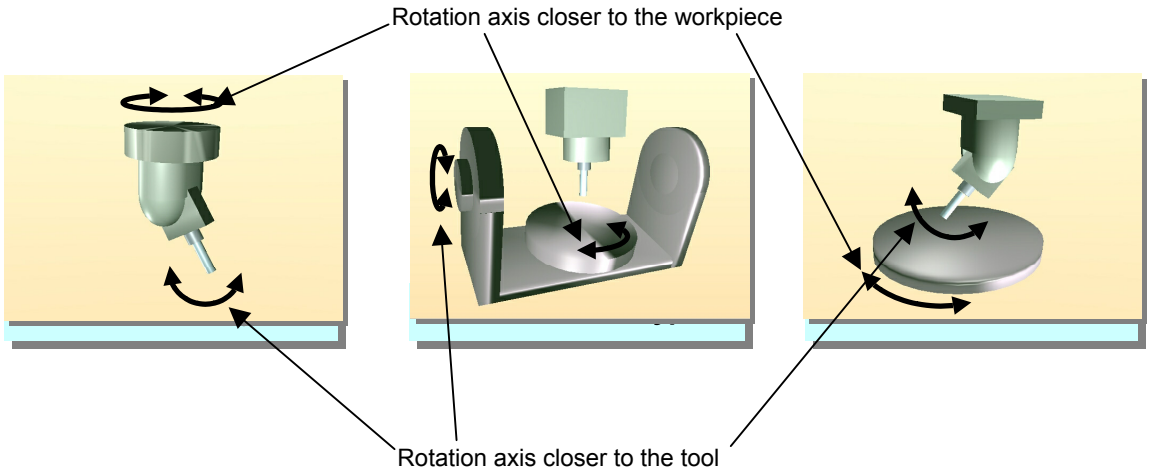
On a 5-axis machine with two rotation axes, one rotation axis functions to tilt the tool toward the workpiece. This rotation axis is referred to as a "rotation axis closer to the tool".

The other rotation axis is referred to as a "rotation axis closer to the workpiece".

Depending on the type of mechanical section, the rotation axis closer to the tool and rotation axis closer to the workpiece are determined as indicated in Table 15.5 (k), “"Rotation axis closer to the tool” and "rotation axis closer to the workpiece"”.

Table 15.5 (k) "Rotation axis closer to the tool” and "rotation axis closer to the workpiece"

Type of mechanical section (parameter No. 19680)	Rotation axis closer to the tool	Rotation axis closer to the workpiece
Tool rotation type (2)	Slave axis	Master axis
Table rotation type (12)	Master axis	Slave axis
Composite type (21)	Tool rotation axis	Table rotation axis



- Singular point and singular point posture on a 5-axis machine

A tool posture is uniquely determined when the angles of the two rotation axes are determined. Usually, however, a combination of the angles of the two rotation axes for achieving a certain tool posture is not determined uniquely.

In particular, such a tool posture that the angle of the rotation axis closer to the workpiece is arbitrary is referred to as a "singular point posture". Moreover, such an angle of the rotation axis closer the tool that the tool posture becomes a singular point posture is referred to as a "singular point (or singular point angle)". When the angle of the rotation axis closer to the tool is a singular point, the center of the rotation axis closer to the workpiece and the tool posture (tool direction) are parallel with each other.

Example:

When the C-axis (about the Z-axis) is the master axis, the B-axis is the slave axis (about the Y-axis), and the reference tool axis direction is along the Z-axis on a 5-axis machine of tool rotation type, the singular point is B = …, 0 deg, 180 deg, ... At this time, an arbitrary C-axis angle represents a singular point posture (in the same direction as the Z-axis or in the direction opposite to the Z-axis).

If the angle of the B-axis is B = 0 (singular point) as shown in Fig. 15.5 (d), for example, an arbitrary C-axis angle represents the same tool posture (singular point posture).

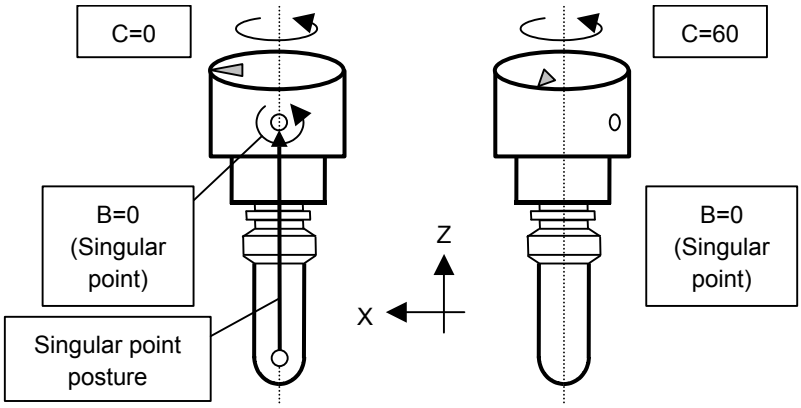


Fig. 15.5 (d) Singular point and singular point attitude

When the reference tool axis direction is inclined (parameters Nos. 19698 and 19699), or the rotation axis is an angular axis (parameters Nos. 19682, 19683, 19687, and 19688), for example, no singular point and singular point posture exist on some machines.

- Conditions to decide that Tool is in singular posture

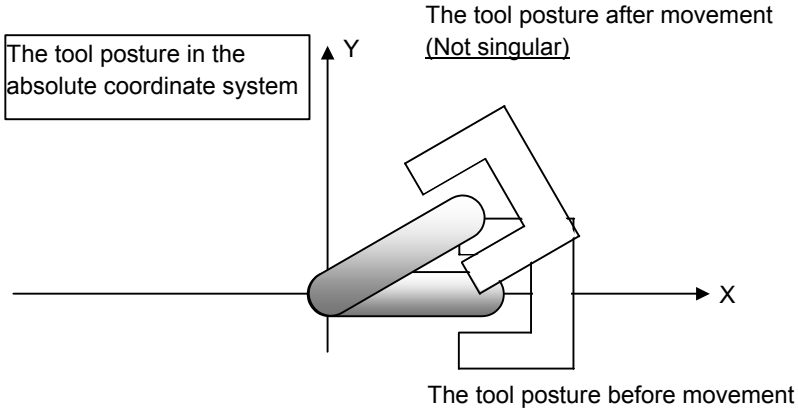
When the angle between the tool and the singular posture is less than the parameter No.11204, it is decided that the tool is in singular posture.
In the descriptions below, the description 'when the tool is in singular…' means that 'when it is decided that the tool is in singular posture….'.
In addition, whether the tool is in singular point posture is decided at each end point of tool paths calculated in the NC. If the parameter No. 11204 value is very small and the feedrate and the override value are large, it is likely that the tool may pass by a singular point without deciding that the tool is in a singular point posture at each end point even when the tool path crosses the singular point. In this case, the tool will not be able to go beyond the singular point. To avoid this problem, set parameter No. 11204 with a value slightly larger than the quotient of the division of the highest rotation axis move speed (the highest cutting feedrate (parameter No. 1432) among the rotation axes or the rapid traverse rate (parameter No. 1420) whichever is higher) by 15000.

- Movement in the case that Tool direction is compensated

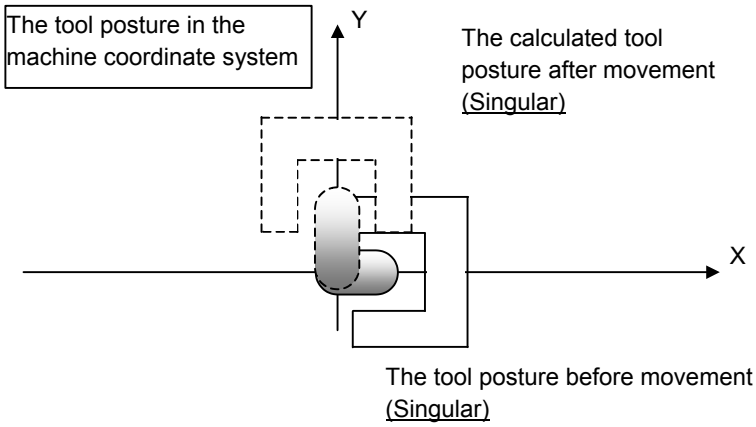
In the case that the tool direction is compensated (bit 0 (RCM) of parameter No.11200=1) on a 5 axis machine, the compensation is performed in every interpolation, and the rotation axes may move to positions which are different from the commanded positions.
In general, there are two pairs of rotation axes positions in the region between 0 degree to 360 degree to turn the tool in a certain direction.
And in the case that the tool is in singular point posture, the position of the rotation axis closer to the workpiece is not determined uniquely.
How the positions are determined is described as follows :

(1) In the case that the current machine position is singular and the position after movement in real time is also singular.
   (a) When the absolute position after movement in real time is not singular, the rotation axis closer to the workpiece does not move.

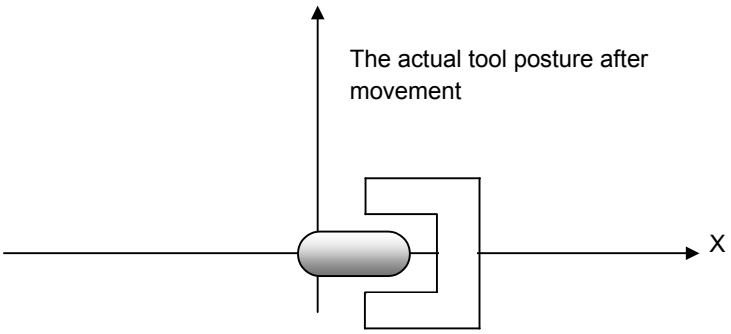
   Example:
   The rotation axis about the Z-axis is the master axis, the rotation axis about the Y-axis is the slave axis, and the reference tool axis direction is along the Z-axis on a 5-axis machine of tool rotation type.



   Assume that the workpiece setting error around the Y-axis exists, and the tool posture after the compensation of tool direction becomes like following figure. (The tools before and after movement are in singular point posture.)



In this case, the rotation axis about the Z-axis (the rotation axis closer to the workpiece) does not move. (Only the rotation axis closer to the tool moves.)

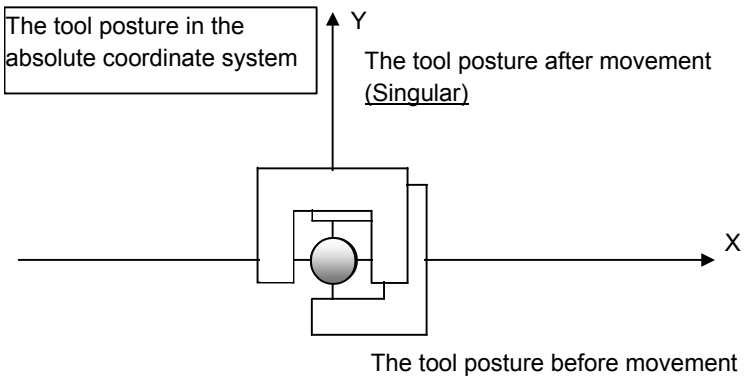


If the parameter No.11204 is set to the value except 0, the tools both before and after movement may be in singular point posture like the above example.
In this case, the rotation axis closer to the workpiece does not move. The rotation axis closer to the tool moves in the move direction in which the tool in the machine system (machine coordinate system) is decided to be in a singular point posture, but not move in the opposite direction.

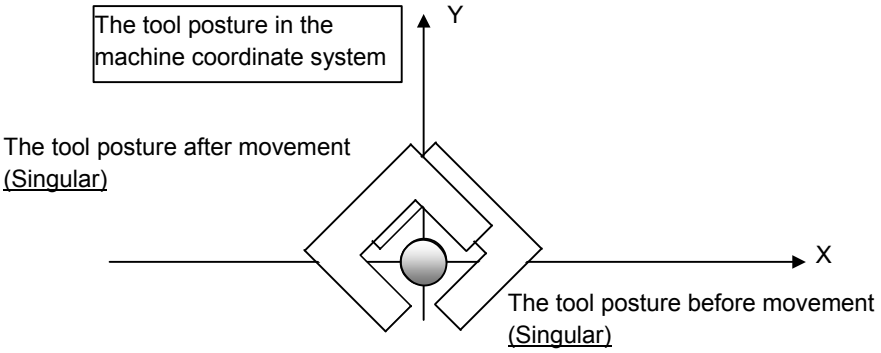
(b) When the absolute position after movement in real time is singular, the rotation axis closer to the workpiece moves as commanded.

Example:

The rotation axis about the Z-axis is the master axis, the rotation axis about the Y-axis is the slave axis, and the reference tool axis direction is along the Z-axis on a 5-axis machine of tool rotation type.



Assume that the workpiece setting error around the Z-axis exists, and the tool posture after the compensation of tool direction becomes like following figure.



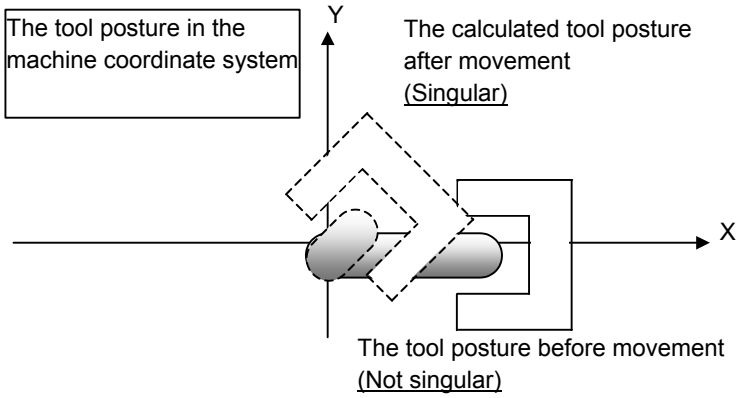
In this case, the rotation axis about Z-axis (the rotation axis closer to the workpiece) moves as commanded.

(2) In the case that the current machine position is not singular and the position after movement in real time is singular.
: The rotation axis closer to workpiece does not move.

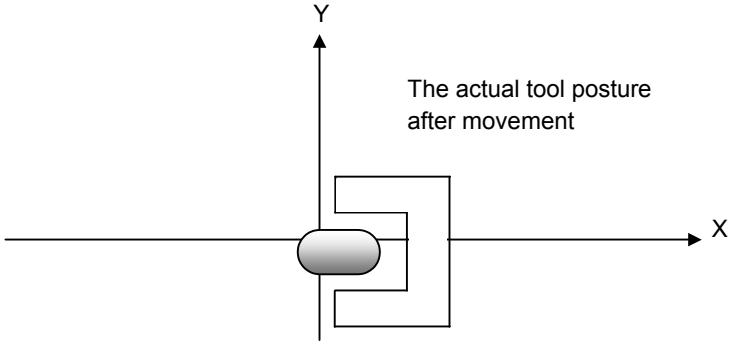
Example:

The rotation axis about the Z-axis is the master axis, the rotation axis about the Y-axis is the slave axis, and the reference tool axis direction is along the Z-axis on a 5-axis machine of tool rotation type.

Assume that the tool posture after the compensation of tool direction becomes like following figure.



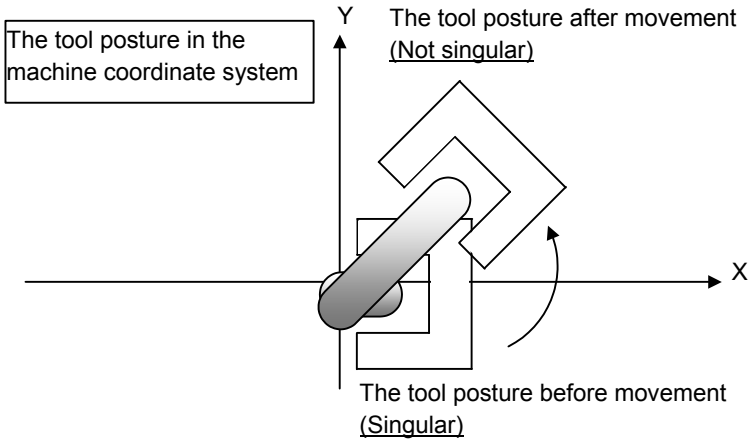
In this case, the rotation axis about the Z-axis (the rotation axis closer to the workpiece) does not move. (Only the rotation axis closer to the tool moves.)



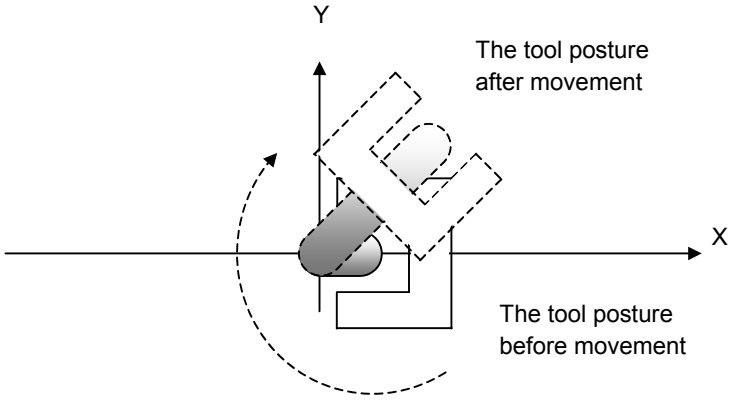
(3) In the case that the current machine position is singular and the position after movement in real time is not singular.
: In order to position the tool to the correct direction, there are two pairs of solutions of rotation axes angles within 0 – 360deg. A solution with which the rotation axis closer to workpiece moves shorter is selected.

Example:
The rotation axis about the Z-axis is the master axis, the rotation axis about the Y-axis is the slave axis, and the reference tool axis direction is along the Z-axis on a 5-axis machine of tool rotation type.

Assume that the tool posture after the compensation of tool direction becomes like following figure.



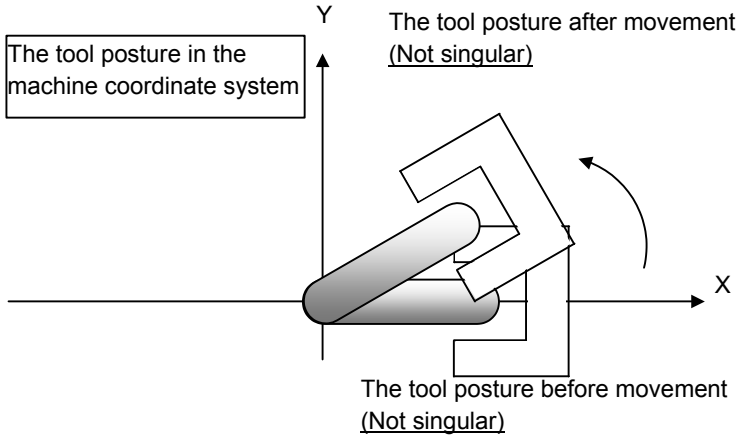
In this case, the rotation axes move to the positions with which the rotation axis about the Z-axis (the rotation axis closer to workpiece) moves shorter. (The tool moves in the direction of the arrow in the above figure.)
The tool does not move in the direction of the arrow in the following figure.



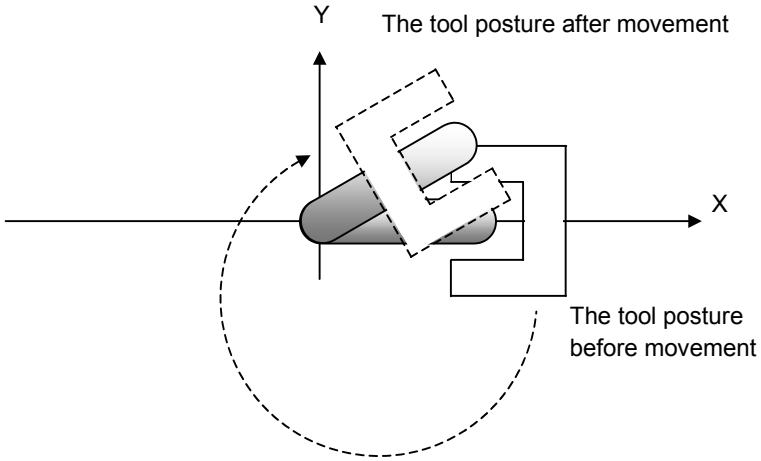
(4) In the case that the current machine position is not singular and the position after movement in real time is not singular.
: In order to position the tool to the correct direction, there are two pairs of solutions of rotation axes angles within 0 – 360deg. One solution with which the rotation axes don't pass the singular point is selected. And, the rotation axes move to the positions of the selected solution.

Example:
The rotation axis about the Z-axis is the master axis, the rotation axis about the Y-axis is the slave axis, and the reference tool axis direction is along the Z-axis on a 5-axis machine of tool rotation type.

Assume that the tool posture after the compensation of tool direction becomes like following figure.



In this case, the rotation axes move to the positions with which the rotation axes don't pass the singular point. (The tool moves in the direction of the arrow in the above figure.)
The tool does not move in the direction of the arrow in the following figure.



- When the tool posture is closer to a singular point posture on a 5-axis machine

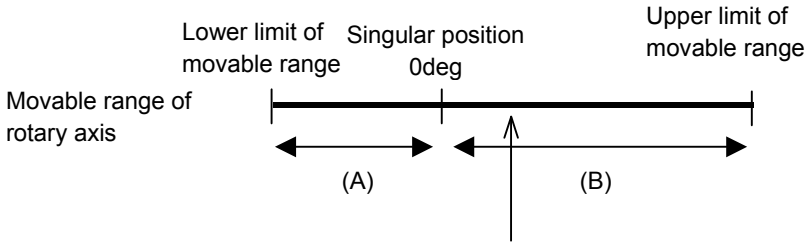
If tool direction compensation is performed on a 5-axis machine, and the tool posture gets closer to a singular point posture during execution of a block, the rotation axis closer to the workpiece may make a large motion.

In addition, if parameter No. 11204 (angle for deciding on a singular point posture) is set with a value, it is likely that the tool direction and tool tip position may be incorrect while the tool is decided to be in a singular point posture. Furthermore, it is likely that the rotation axis closer to the tool may abruptly move by the parameter-set angle. So, do not set parameter No. 11204 with a value larger than necessary.

- Notes in the case that rotary axes have movable range

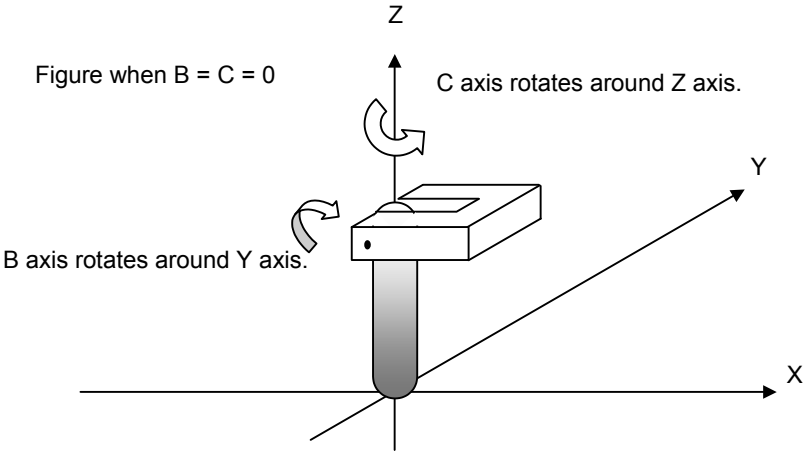
When the tool direction is compensated on a 5-axis machine, there can be the case that the machine position does not pass the singular posture during a progress of work at all.
In this case, in accordance with 4) described above, the solution with which the rotary axes don't pass the singular point is always selected. Therefore, in this case, the movement direction(area) depends on where the tool exists in comparison with the singular point, because the tool position is always selected so that the rotary axes don't pass the singular point.

In the case that rotary axes have movable range and a singular point exists in that range, Workpiece setting error compensation must be activated after the rotary axes have been moved to the range where the rotary axes should move, that is, the range (A) between the lower limit and the singular point or the range(B) between the upper limit and the singular point.
Generally speaking, if (A) and (B) are different, it is recommended that the larger one is selected.

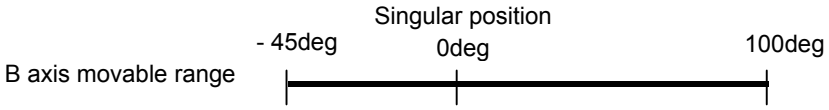


Workpiece setting error compensation must be activated after the rotary axes have been moved to the range where the rotary axes should move

The following is an example that the movement direction(area) depends on position of the rotary axes before starting workpiece setting error compensation.
The Master axis is C axis around Z axis, and the Slave axis is B axis around Y axis(when C=0). Tool direction is +Z when B=C=0.
Here, suppose B axis' movable range is –45 deg to +100 deg.

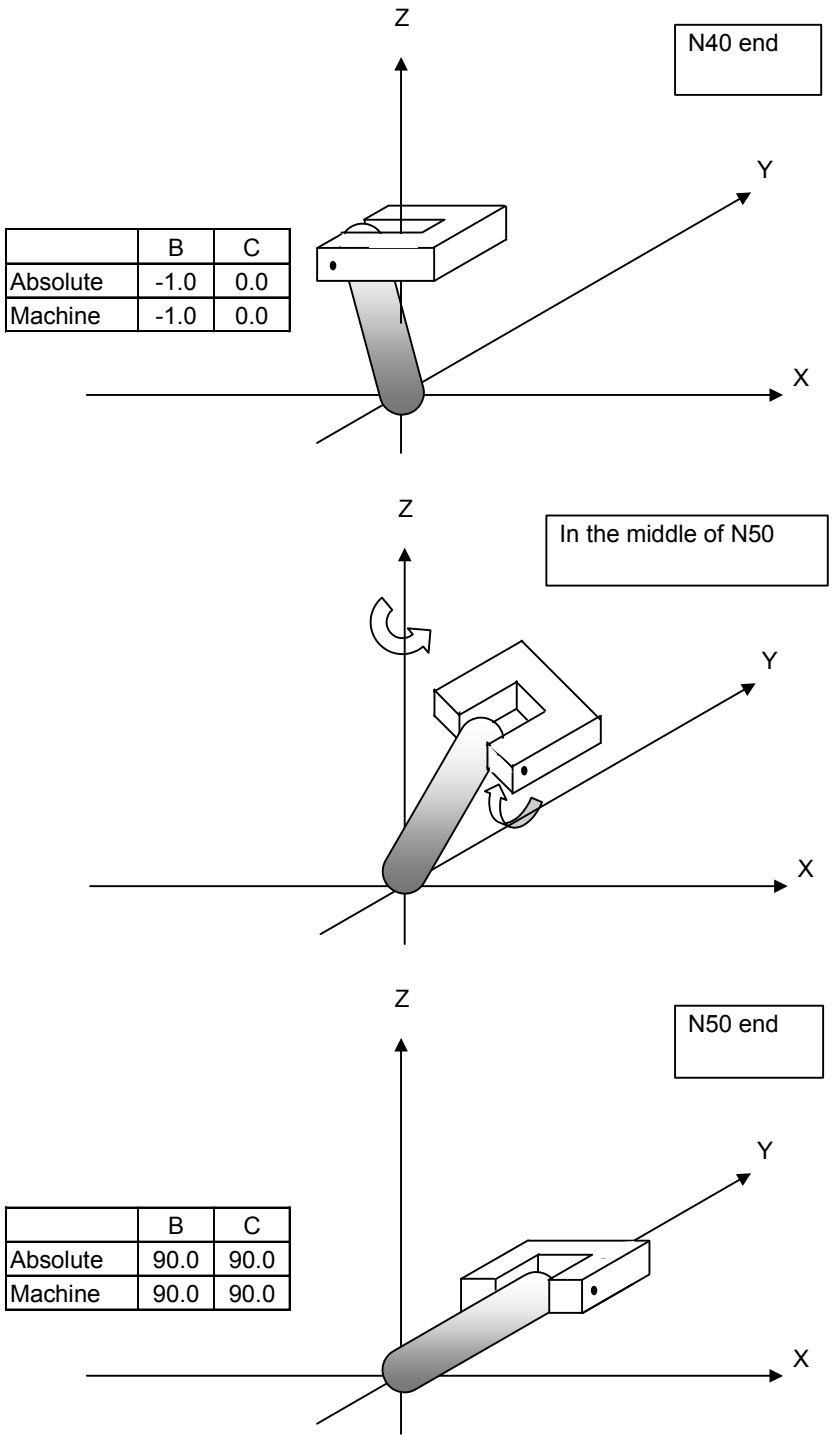


When B=0, the tool posture does not change even if C axis moves. That means the position B=0 is singular point.



Firstly, suppose the following program without Workpiece setting error.

Program without Workpiece setting error
O1
N10 G5.1 Q1
N20 G90 G01 B-1.0 C0 F1000
N30 G43.4 H1
N40 X0 Y0 Z0
N50 B90.0 C90.0



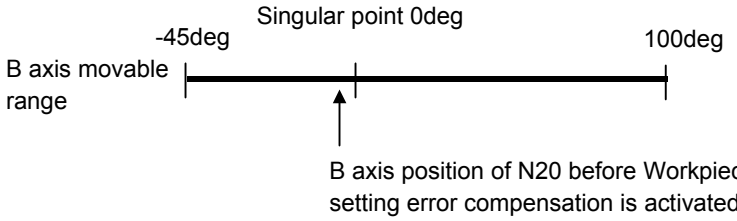
	B	C
Absolute	-1.0	0.0
Machine	-1.0	0.0

	B	C
Absolute	90.0	90.0
Machine	90.0	90.0

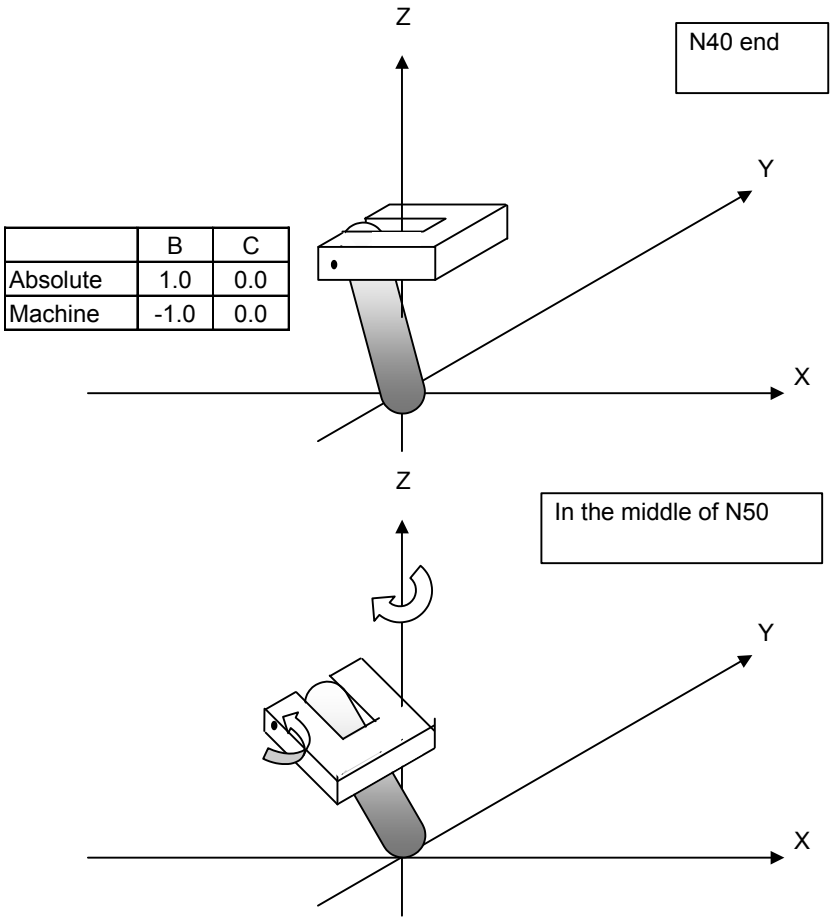
At N50, machine position moves to B90.0 and C90.0, as commanded.

Next, suppose there is the error -2.0deg around Y axis and Workpiece setting error Δb=-2.000 is set, and the block N25 is added as follows :

On O2, B axis position is –1.0 before Workpiece setting error compensation is activated, which is between the lower limit of movable range and the singular point of B axis.



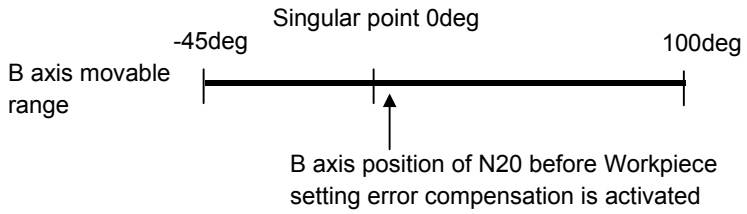
O2
N10 G5.1 Q1
N20 G90 G01 B-1.0 C0 F1000 ; B axis machine position is between the lower limit and the singular point
N25 G54.4 P1
N30 G43.4 H1
N40 X0 Y0 Z0
N50 B90.0 C90.0



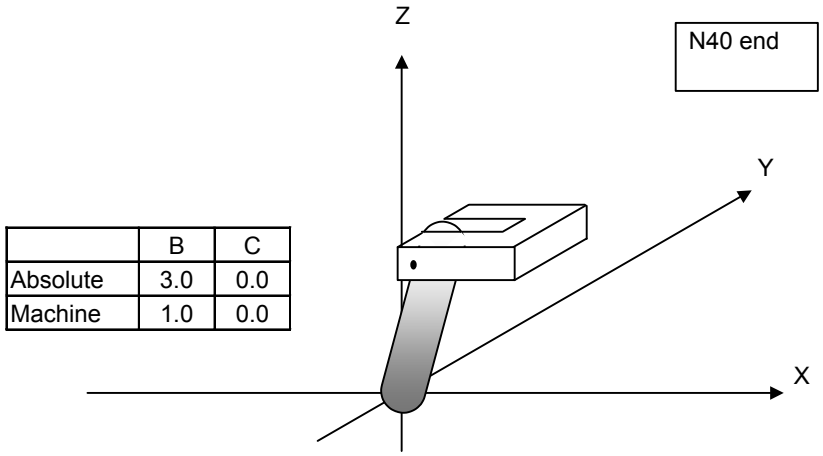
	B	C
Absolute	1.0	0.0
Machine	-1.0	0.0

During N50, Machine position does not pass the singular point at all. So, B axis moves from the starting position(–1.0) to the position(B–90.0,C–90.0). As the result, B axis moves over the lower limit of B axis movable range.

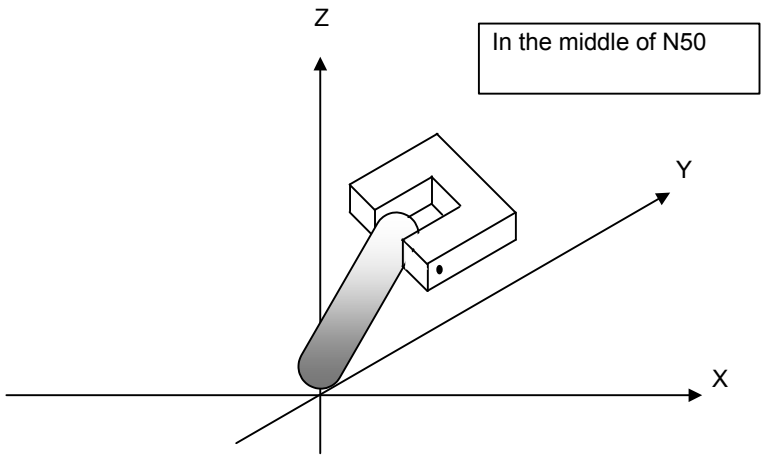
Therefore, suppose the program O3 in which B axis position is changed to 1.0, which is between the singular point and the upper limit, before Workpiece setting error compensation is activated.
Then, the range between the singular point and the upper limit of B axis movable range is the range where B axis should move.

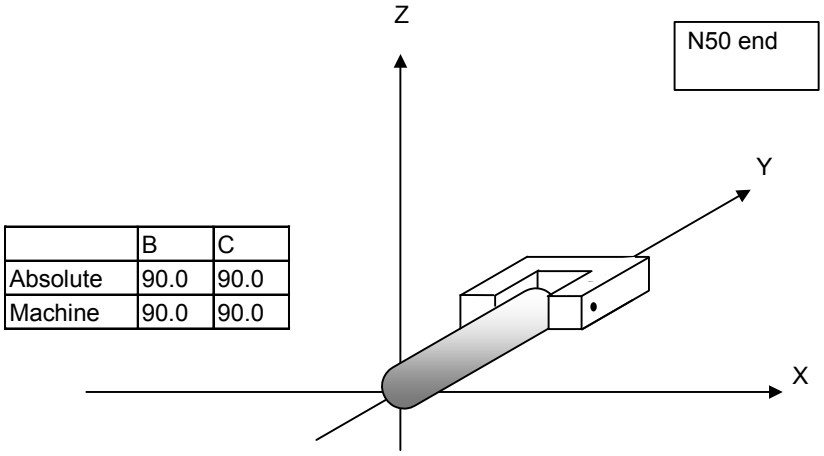



```
O3
N10 G5.1 Q1
N20 G90 G01 B1.0 C0 F1000 ; B axis machine position is between the upper limit and the singular point
N25 G54.4 P1
N30 G43.4 H1
N40 X0 Y0 Z0
N50 B90.0 C90.0
```



	B	C
Absolute	3.0	0.0
Machine	1.0	0.0





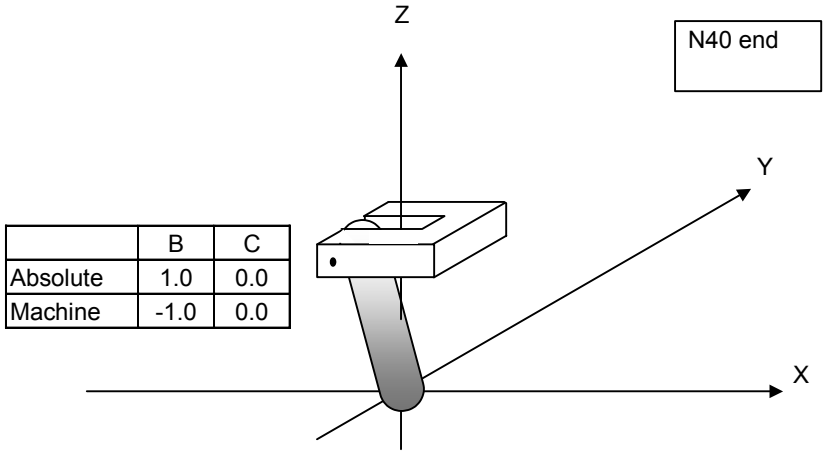
	B	C
Absolute	90.0	90.0
Machine	90.0	90.0

This time, machine position moves to B90.0,C90.0 during N50. As the result, B axis does not move over the lower limit of B axis movable range.

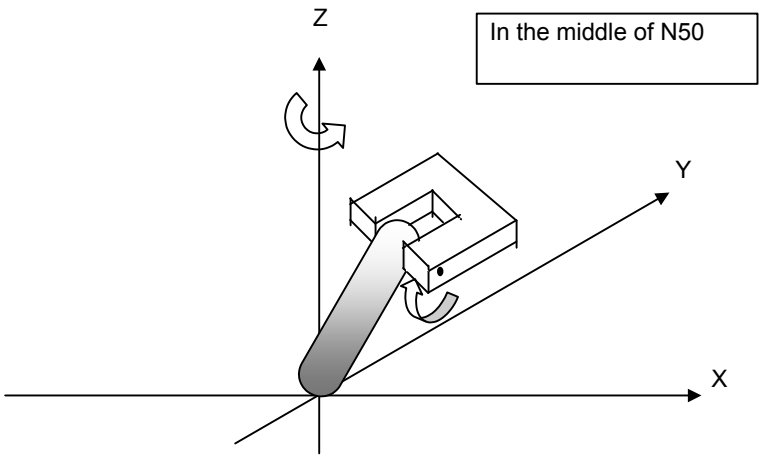
In O2, the case that B axis moves over the limit of movable range is the case that machine position does not make the singular point posture during N50.
When parameter No.11204 has 0, whether the tool posture is a singular point posture or not is judged strictly. So when the tool posture is almost a singular point posture, the tool posture is not regarded as a singular point posture.
When parameter No.11204 has a proper value and the tool posture in the machine coordinate system is regarded as a singular point posture, there is the possibility that the position passes the singular point.
In the above example, when parameter No.11204 has 0.1, the machine position come to make the singular point posture during N50 in O2, and B axis passes the singular point and moves to B90.0,C90.0 during N50.

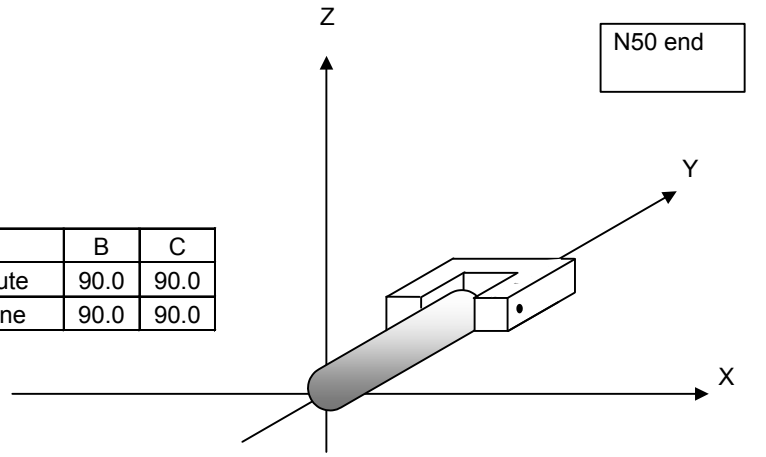
```
O2
N10 G5.1 Q1
N20 G90 G01 B-1.0 C0 F1000 ;     B axis is between the lower limit and the singular point.
N25 G54.4 P1
N30 G43.4 H1
N40 X0 Y0 Z0
N50 B90.0 C90.0
```

O2 is executed with the parameter No.11204=0.1.



	B	C
Absolute	1.0	0.0
Machine	-1.0	0.0





	B	C
Absolute	90.0	90.0
Machine	90.0	90.0

At N50, the machine position moves to B90.0,C90.0. As the result, B axis does not move over the lower limit of B axis movable range.

- Error for which the singular point processing is unnecessary

Even if tool direction compensation must be performed in a 5-axis machine, the above mentioned singular point processing becomes unnecessary if all workpiece setting errors in rotation directions are 0 (only X, Y, and Z directions) or there is only an error about the rotation axis closer to the workpiece.
In this case, rotation axis compensation is performed simply by shifting the rotation axis closer to the workpiece by a distance corresponding to the error; the above mentioned singular point processing is not carried out.

[Tool rotation type example]

In a tool rotation type, specify the C-axis (rotating about the Z-axis) and B-axis (rotating about the Y-axis), respectively, as the master and slave rotation axes.
The C-axis is a rotation axis closer to the workpiece.
No singular point posture processing is carried out if the workpiece setting errors in rotation directions are:

Rotation direction error Δa (rotation error about the X-axis): 0°
Rotation direction error Δb (rotation error about the Y-axis): 0°
Rotation direction error Δc (rotation error about the Z-axis): α°

The B-axis will move as directed. The C-axis will move to a position compensated by α° with respect to the specified position.

[Table rotation type example]

In a table rotation type, specify the A-axis (rotating about the X-axis) and C-axis (rotating about the Z-axis), respectively, as the master and slave rotation axes.
The C-axis is a rotation axis closer to the workpiece.
No singular point posture processing is carried out if the workpiece setting errors in rotation directions are:

Rotation direction error Δa (rotation error about the X-axis): 0°
Rotation direction error Δb (rotation error about the Y-axis): 0°
Rotation direction error Δc (rotation error about the Z-axis): α°

The A-axis will move as directed. The C-axis will move to a position compensated by α° with respect to the specified position.

[Composite type example]

In a composite type, specify the B-axis (rotating about the Y-axis) and C-axis (rotating about the Z-axis), respectively, as the tool and table rotation axes.
The C-axis is a rotation axis closer to the workpiece.
No singular point posture processing is carried out if the workpiece setting errors in rotation directions are:

Rotation direction error Δa (rotation error about the X-axis): 0°
Rotation direction error Δb (rotation error about the Y-axis): 0°
Rotation direction error Δc (rotation error about the Z-axis): α°

The B-axis will move as directed. The C-axis will move to a position compensated by α° with respect to the specified position.

- Tool direction compensation is not performed

When tool direction compensation is not performed (bit 0 (RCM) of parameter No. 11200 is set to 0), rotation axes move to the position that is commanded by operator. Therefore, excessive motion of rotation axes in singular point generated when there is a workpiece setting rotation error that tilt the workpiece can be evaded.

This example uses following settings.

Workpiece setting error
Δa = 0.0
Δb = 10.0
Δc = 0.0

```
O0001
N01G90G00X0Y0Z200.0B0C0
N02G5.1Q1
N03G54.4P1
N04X0Y0Z300.0B0C0
N05G43H1
N06G01X200.0Y0F100.0
N07X200.0Y200.0
```

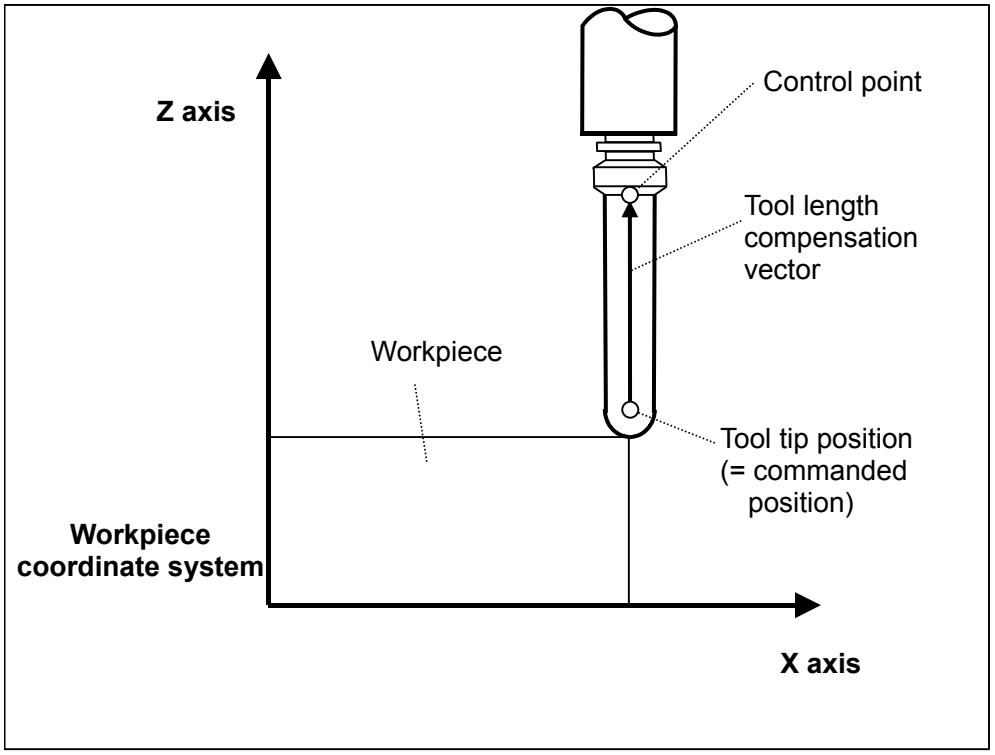


Fig.15.5 (e) Tool tip cutting when there is no workpiece setting error

If tool direction compensation is performed, rotation axes are compensated to match direction of the tool against the workpiece to direction of the tool when there is no workpiece setting error. (Fig.15.5 (f))

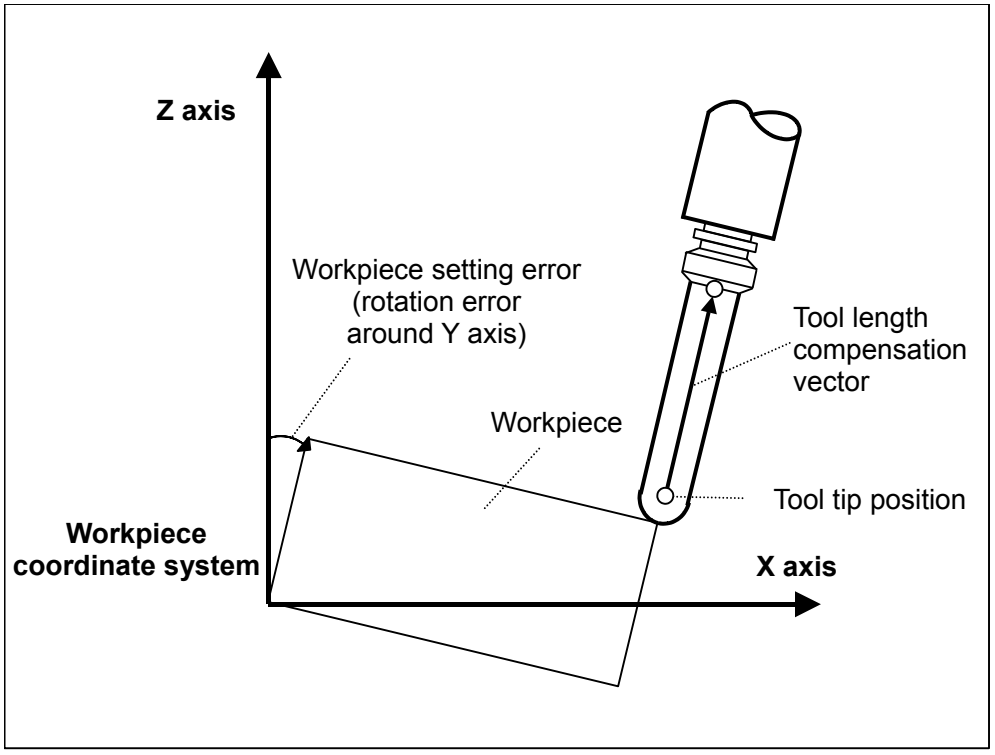


Fig.15.5 (f) Tool tip cutting when there is workpiece setting error (When RCM=1)

If tool direction compensation is not performed, the tool tip position against the workpiece matches the tool tip position when there is no workpiece setting error, and positions of rotation axes matches positions of rotation axes when there is no workpiece setting error. Therefore, even if there is workpiece setting

error, the position of tool tip cutting is correct and the tool direction on workpiece coordinate system matches tool direction when there is no workpiece setting error. (Fig.15.5 (g))

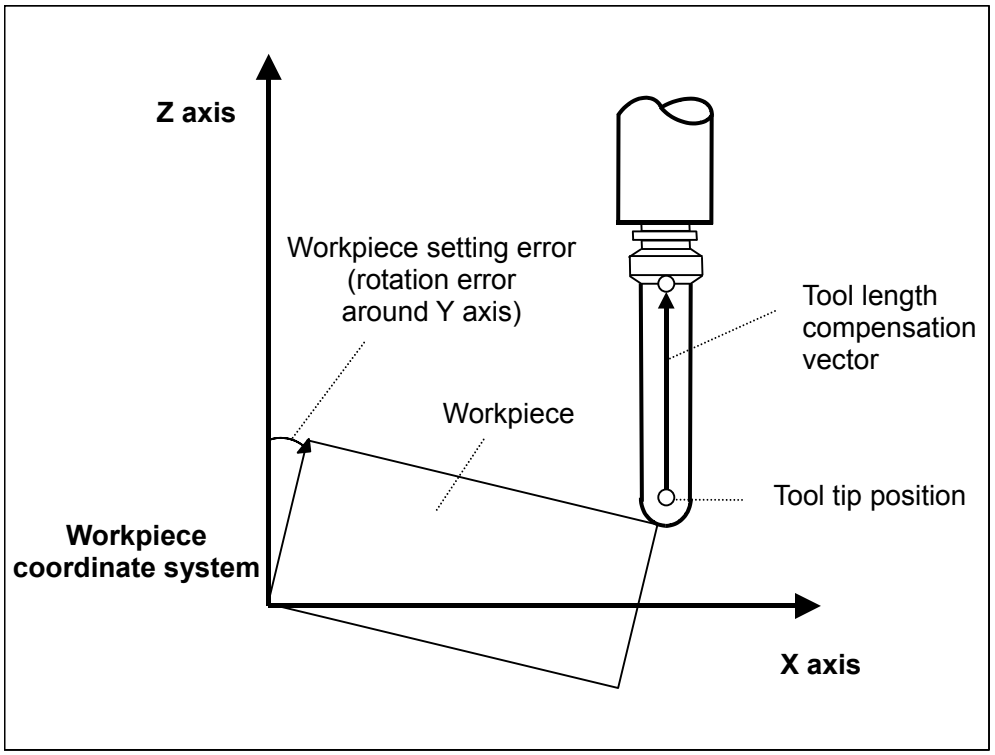


Fig.15.5 (g) Tool tip cutting when there is workpiece setting error (When RCM=0)

When tool length compensation is not active:
When tool length compensation mode or tool center point control mode, tool tip position (command position) is compensated. In case of other, tool holder tip position is compensated as the tool holder tip position against the workpiece matches the tool holder tip position when there is no workpiece setting error. For the definition of tool holder tip position, refer to Fig.15.5 (h).

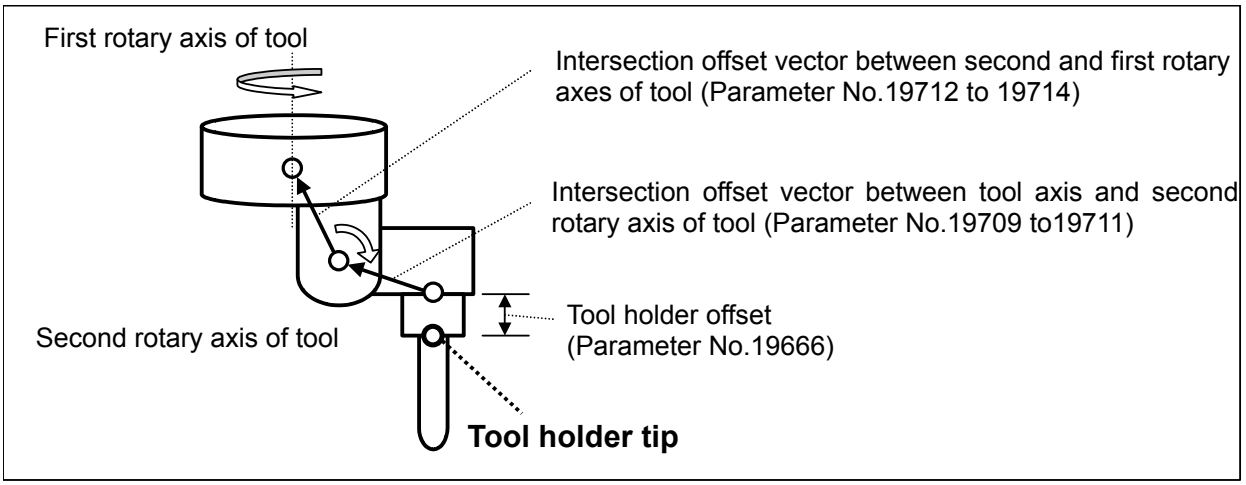


Fig.15.5 (h) Definition of tool holder tip position

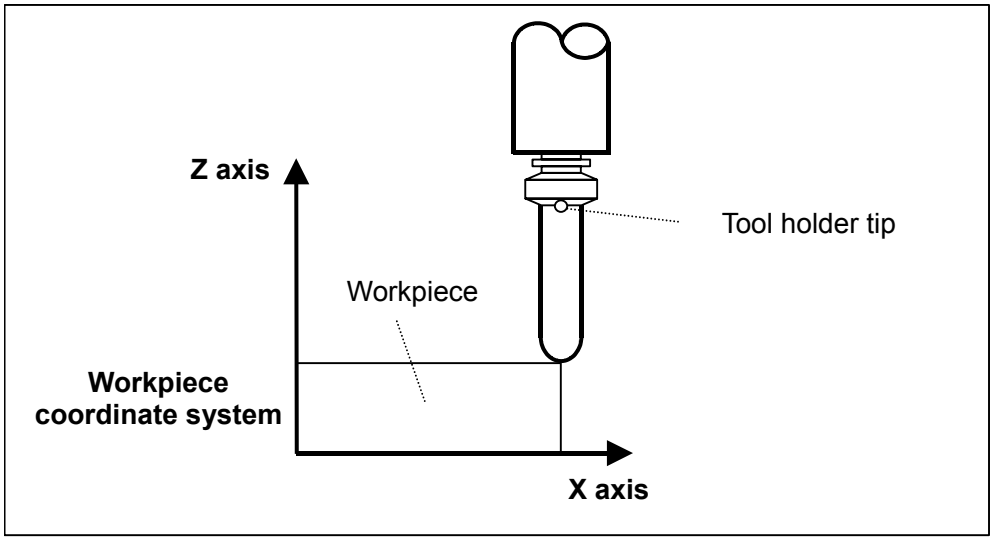


Fig.15.5 (i) When there is no workpiece setting error

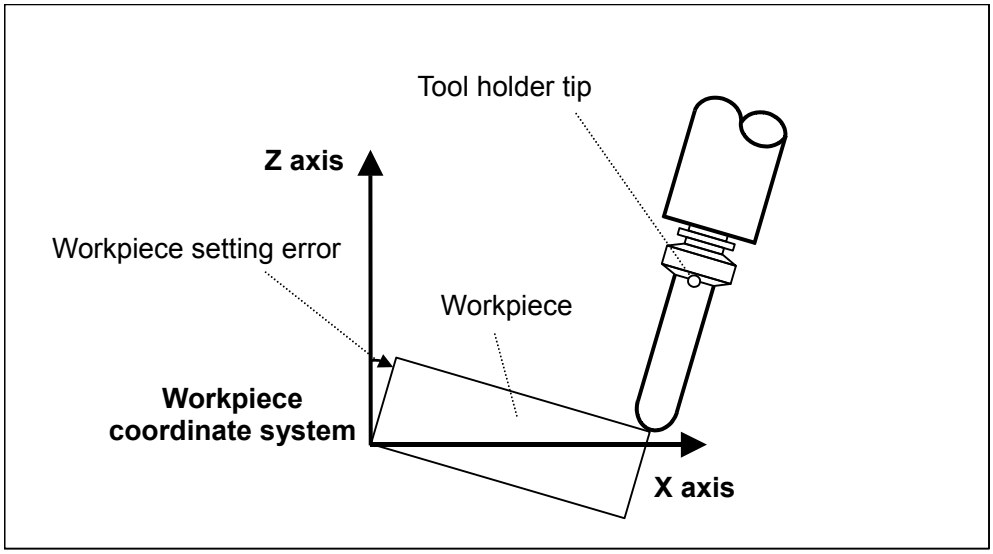


Fig.15.5 (j) When there is workpiece setting error (When RCM=1)

When tool direction compensation is not performed (bit 0 (RCM) of parameter No. 11200 is set to 0) and tool length compensation is no effect, tool holder tip position is same as the tool holder tip position when there is no workpiece setting error. (Fig.15.5 (k)) In this case, the tool tip position might not match the tool tip position when there is no workpiece setting error, because tool direction compensation is not performed.

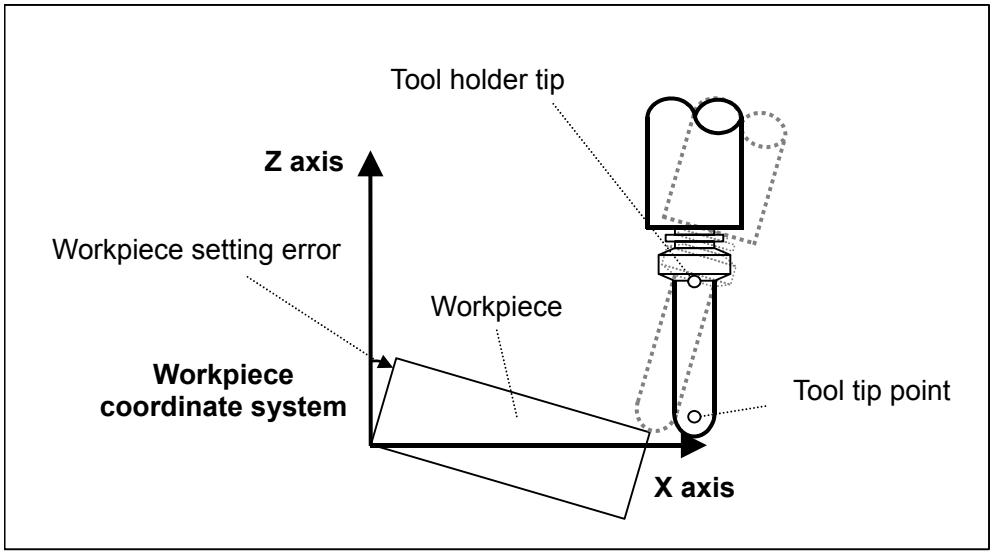


Fig.15.5 (k) When there is workpiece setting error (When RCM=0)

When tool cutter compensation mode:
As the above-mentioned, when tool length compensation mode or tool center point control mode, tool tip position is compensated, and in other case, tool holder tip position is compensated.
Therefore, when tool direction compensation is not performed (bit 0 (RCM) of parameter No. 11200 is set to 0), the program command position during tool cutter compensation might differ from shape of workpiece in actual machining. (Fig.15.5 (n))

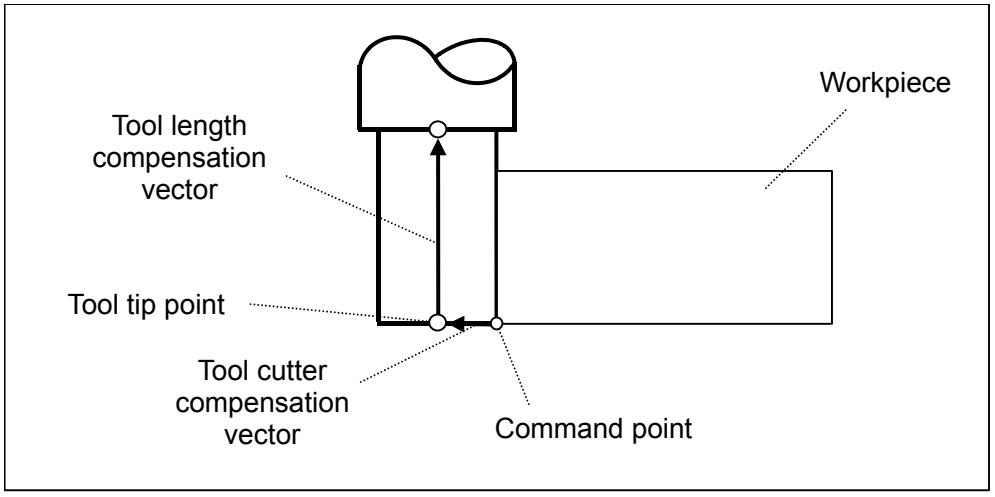


Fig.15.5 (l) Tool cutter compensation when there is no workpiece setting error

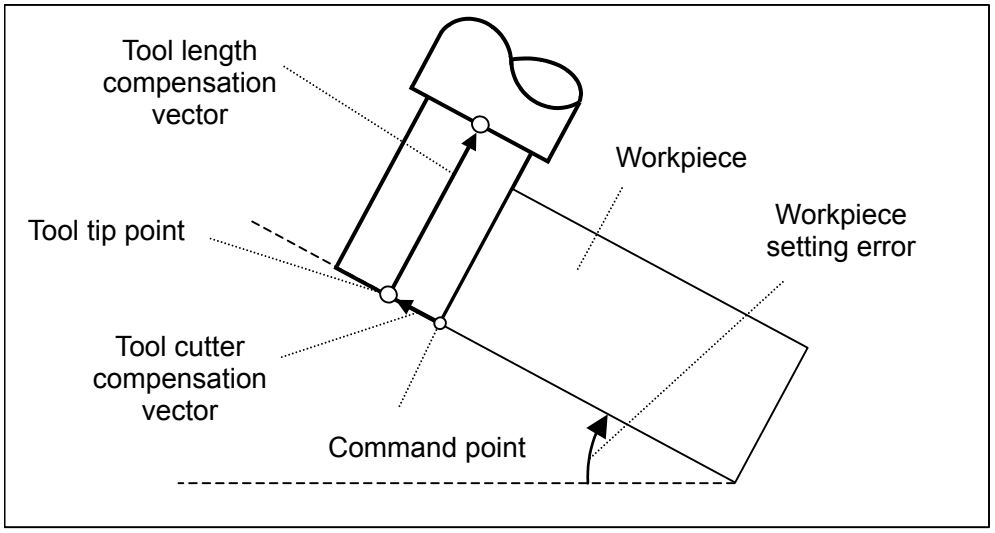


Fig.15.5 (m) Tool cutter compensation when there is workpiece setting error (When RCM=1)

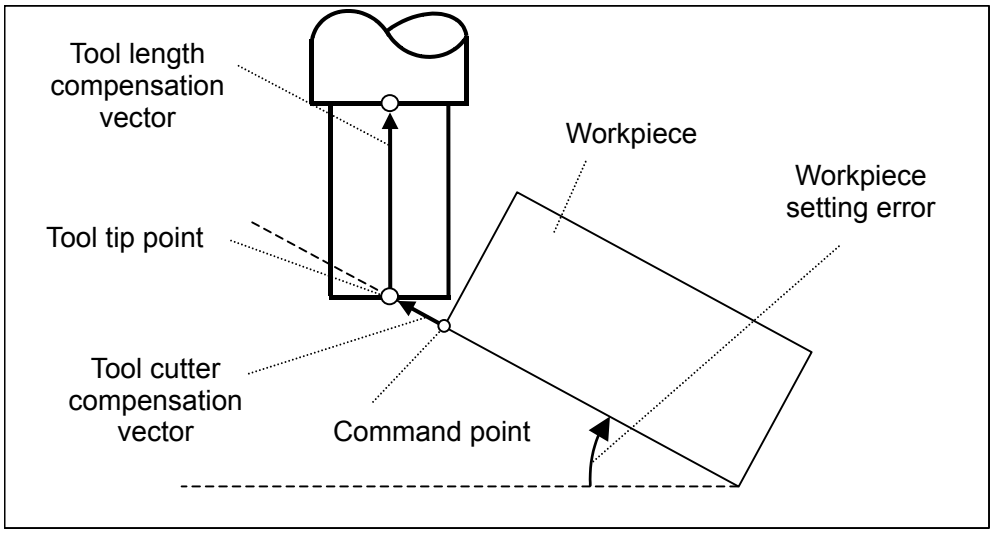


Fig.15.5 (n) Tool cutter compensation when there is workpiece setting error (When RCM=0)

When there is no workpiece setting rotation error that tilt the workpiece (example: Fig.15.5 (p)), converted command point matches shape of the workpiece. (Above difference doesn't occur.)

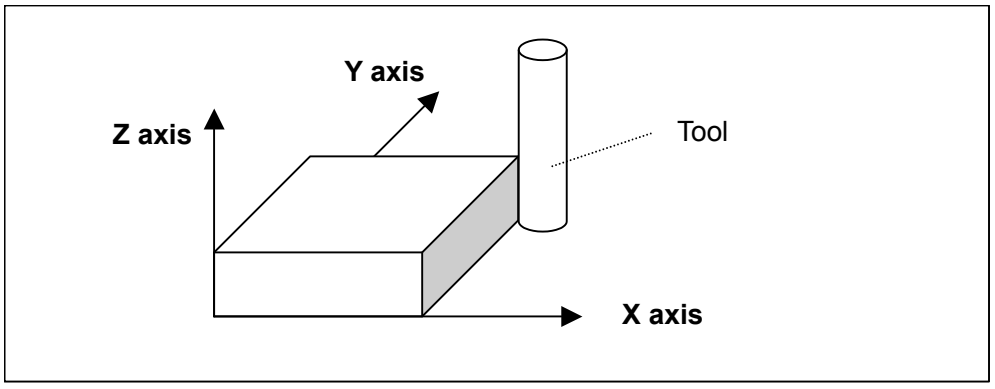


Fig.15.5 (o) Tool cutter compensation when there is no workpiece setting error

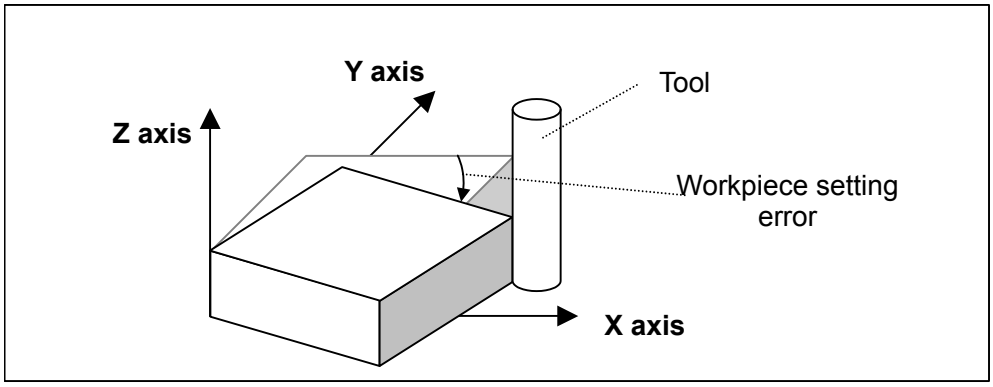


Fig.15.5 (p) Tool cutter compensation when there is only workpiece setting error around Z axis (When RCM=0)

When drilling is performed, or the workpiece is machined with the side of a tool:
When tool direction compensation is not performed, the tool direction against the workpiece is changed. In this case, a tool might interfere with the workpiece or depart from the workpiece as much as workpiece setting error. (This compensation type is not suited when drilling is performed or the workpiece is machined with the side of a tool.)

This example uses following settings.

Workpiece setting error
$\Delta a = 0.0$
$\Delta b = 10.0$
$\Delta c = 0.0$

```
O0002
N01G90G00X0Y0Z200.0B0C0
N02G5.1Q1
N03G54.4P1
N04X0Y0Z200.0B0C0
N05G43H1
N06G01X200.0Y0F100.0
N07X200.0Y200.0
```

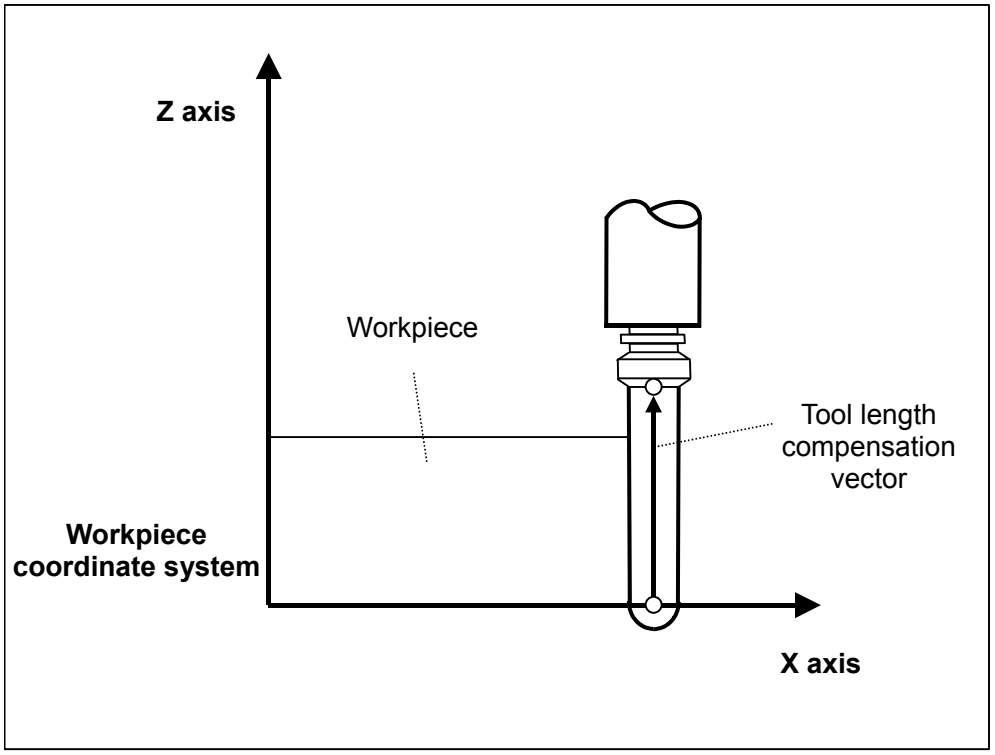


Fig.15.5 (q) The workpiece is machined with the side of a tool when there is no workpiece setting error

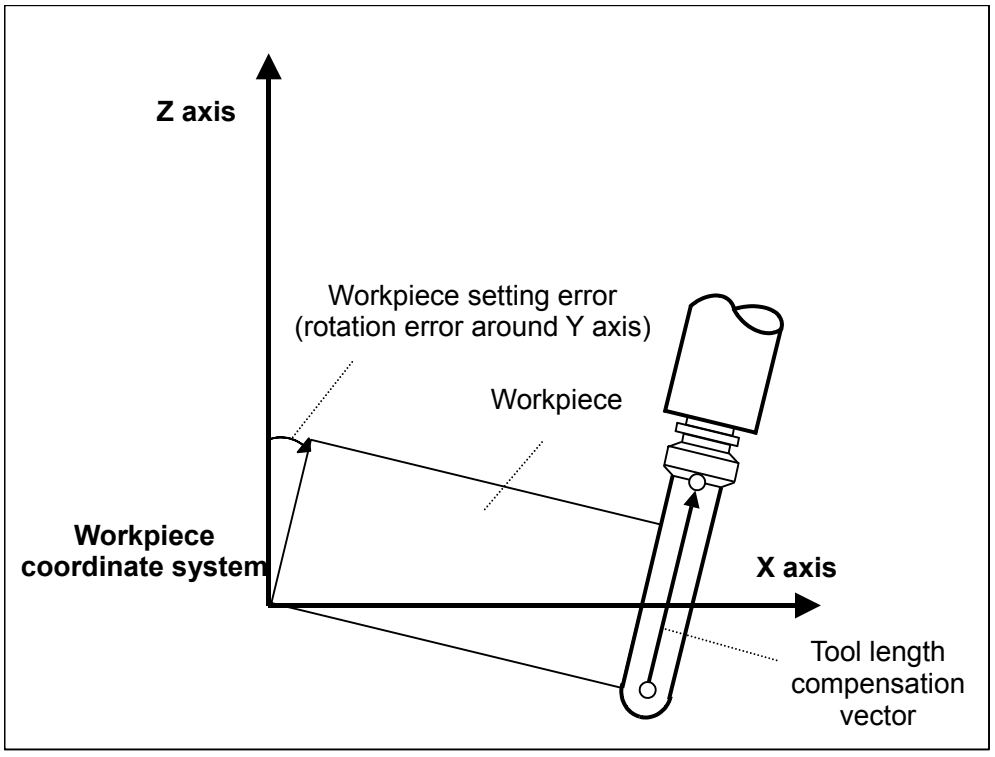


Fig.15.5 (r) The workpiece is machined with the side of a tool when there is workpiece setting error (When RCM=1)

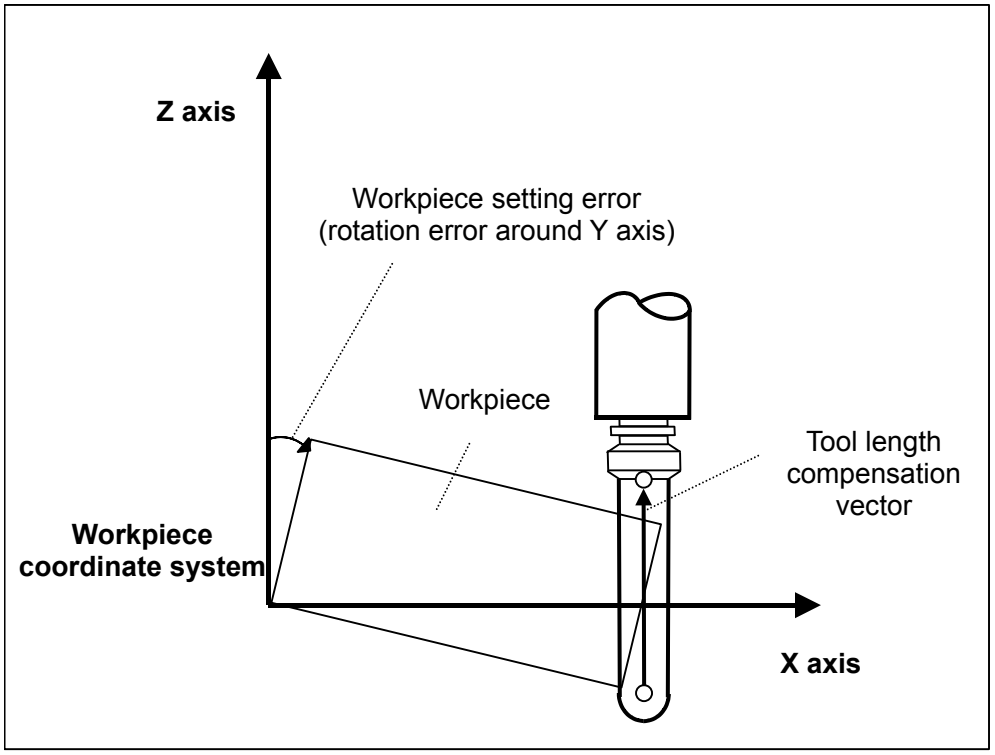


Fig.15.5 (s) The workpiece is machined with the side of a tool when there is workpiece setting error (When RCM=0)

- Absolute position display

Whether absolute coordinates in the workpiece setting error mode are to be displayed in the workpiece coordinate system or workpiece setting coordinate system can be chosen by using bit 6 (DAK) of parameter No. 3106.

- Distance-to-go display

During workpiece setting error compensation, distance-to-go display shows the distance to go along the axis along which the tool actually moves, not the distance in the workpiece setting coordinate system. If a workpiece setting error in the rotation direction is set, the tool posture may become a singular point posture. (For details, see the section on "Singular point and singular point posture on a 5-axis machine".)
The machine coordinate on the rotation axis at this time is determined through singular point processing. Depending on the angle solution for the rotation axis that is selected, the distance to go may be recalculated.

- Custom macro variables

To system variables #5041 through #5048 (current position on each axis), the coordinates in the workpiece coordinate system are assigned.

- Range of rotary axes movements

In the case that Tilted working plane indexing or Type II of Tool center point control or Type II of Tool radius compensation for 5-axis machining is used, there are the parameters Nos.19741 to 19744 to limit the range of rotary axes movements. When tool direction is compensated (bit 0 (RCM) of parameter No.11200=1), the rotary axes are compensated after the range of rotary axes movements are limited. Therefore, when the rotary axes are compensated, the rotary axes positions may be out of the range of rotary axes movements.
Therefore, in the case that the rotary axes have the range of movements, set the stored stroke limit check to prevent a movement over the range.

- Tolerance for assuming rotation direction errors to be 0

Tolerance for assuming rotation direction errors to be 0 can be set in parameters Nos. 1750 to 11752.

When the machine has a table rotation axis

When the machine has a table rotation axis, parameters Nos. 11750 to 11752 are valid for rotation direction errors Δa, Δb, and Δc when the table rotation axis position in the workpiece coordinate system is 0.

Parameters Nos. 11750 to 11752 have the following meanings:

Parameter No. 11750: When workpiece setting error Δa is less than or equal to this value, the error is assumed to be 0.

Parameter No. 11751: When workpiece setting error Δb is less than or equal to this value, the error is assumed to be 0.

Parameter No. 11752: When workpiece setting error Δc is less than or equal to this value, the error is assumed to be 0.

When the absolute value of the rotation direction error when the table rotation axis position in the workpiece coordinate system is 0 is less than or equal to the setting of parameters Nos. 11750 to 11752, the rotation direction error is assumed to be 0. (The least input increment for rotation direction errors is determined by the setting of parameter No. 11201.)

An example of the table rotation type is shown below.

Axis configuration: X, Y, Z, B, C

Workpiece coordinate system offset: All 0 for rotation axes

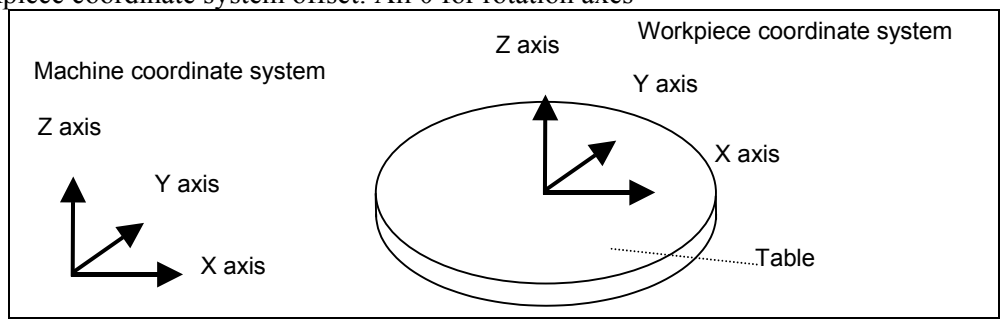


Fig. 15.5 (t) Table position when B = 0.0

Workpiece setting error compensation: Δx=0, Δy=0, Δz=0, Δa=0, Δb=0, Δc=0.5, B=90.0, C=0

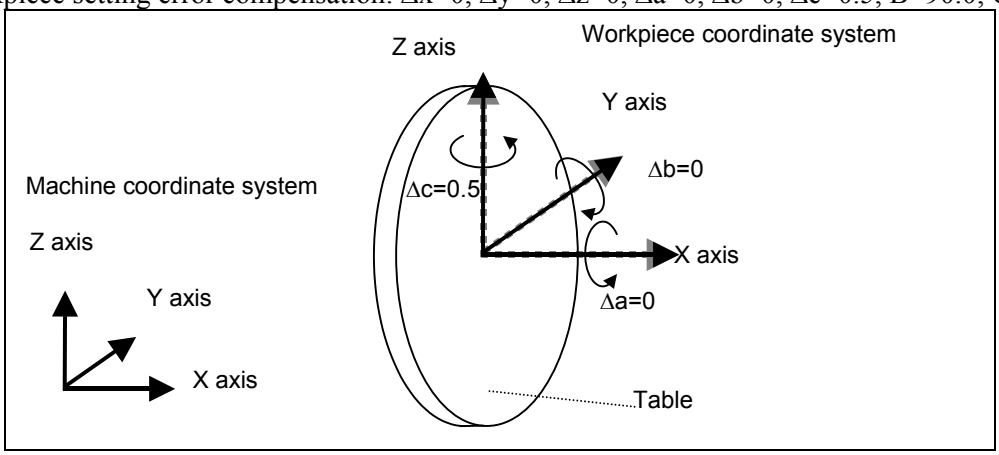


Fig. 15.5 (u) Table position when a workpiece setting error is set (B = 90.0)

Since B = 90.0 is set for the table rotation axis position when workpiece setting errors are measured, when the position about the B-axis in the machine coordinate system is 90.0, workpiece setting errors Δa, Δb, and Δc are defined as shown in Fig. 15.5 (u).

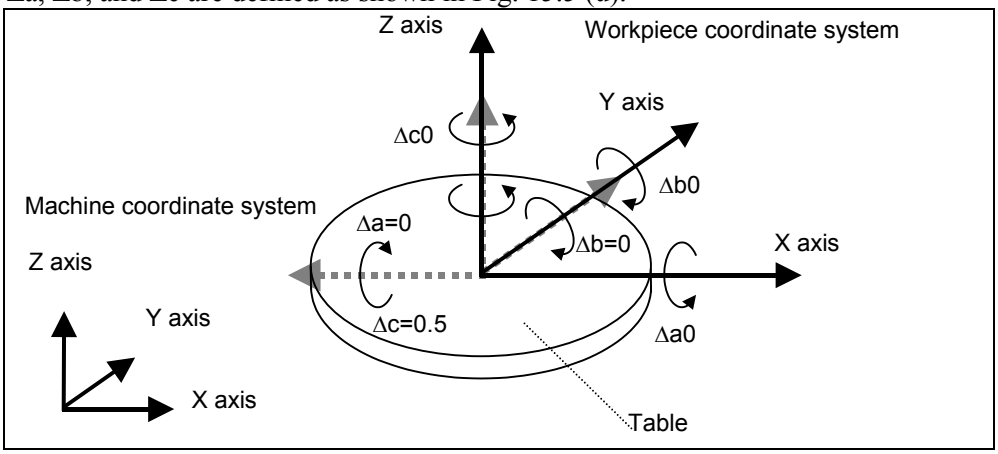


Fig. 15.5 (v) Workpiece setting error (B=0.0)

When the position about the B-axis in the workpiece coordinate system is set to 0.0, the table rotates by the workpiece setting errors as shown in Fig. 15.5 (v). Δa0, Δb0, and Δc0 are the rotation direction errors about the X-, Y-, and Z-axes in the workpiece coordinate system when the table rotation axis position is set to 0 in the workpiece coordinate system. For the case shown in Fig. 15.5 (v), Δa0 is in the direction opposite to that of Δc and Δc is 0.5, Δb is 0.0 for Δb0, and Δa is 0.0 for Δc0, so they are assumed as follows:

Δa0 = -0.5, Δb0 = 0.0, Δc0 = 0.0

The settings of parameters Nos. 11750 to 11752 are valid for these Δa0, Δb0, and Δc0.

Accordingly, Δa0=0.0 is assumed to be 0 when the parameters are set as follows:

Parameter No. 11201=0
Parameter No. 11750=500
Parameter No. 11751=0
Parameter No. 11752=0

When the parameters are set as follows: Δa0 remains -0.5:

Parameter No. 11201=0
Parameter No. 11750=409
Parameter No. 11751=0
Parameter No. 11752=500

> ⚠ **CAUTION**
> Note that the following equations may not be satisfied depending on the workpiece setting errors:
> Δ a 0 = -Δc, Δb0 = Δb, Δc0 = Δa
> The table rotates for Δa0, Δb0, and Δc0, and Δa, Δb, and Δc in the following order:
> a → b → c
> For this reason, when the table rotates about rotation axes, the rotation order (from a to b to c) is changed.
> For Δa0 and Δc, different signs are used because they are different in the rotation direction. Accordingly, when Δa or Δb has a value, the above equations are not satisfied.

- Upper and lower limits of rotation direction errors

Upper and lower limits of rotation direction errors can be set in parameters Nos. 11753 to 11758.

If a rotation direction error is outside the range set in the corresponding parameters Nos. 11753 to 11758, alarm PS0517, “SETTING ERROR AMOUNT IS OUT OF RANGE” is issued when workpiece setting error compensation is started.
When the machine has table rotation axes, the settings of parameters Nos. 11753 to 11758 are valid for rotation direction errors Δa, Δb, and Δc when the table rotation axis position in the workpiece coordinate system is 0.

- Skip function

During workpiece setting error compensation, the skip function (G31) can be specified. The coordinate system of skip positions that store system variables is changed as Table15.5 (l) by setting of bit 5 (LV3) of parameter No. 5400 and bit 3 (WSK) of parameter No. 11200.

Table15.5 (l) Coordinate system of skip positions during workpiece setting error compensation

System variables	Coordinate system of skip positions			
	LV3=0 WSK=0	**LV3=1 WSK=0**	**LV3=0 WSK=1**	**LV3=1 WSK=1**
#5061 - #5080 #100151 - #100200	Workpiece coordinate system (machine system) (B)	Workpiece coordinate system (machine system) (B)	Workpiece coordinate system (command system) (C)	Feature coordinate system (command system) (D)
#151001 - #151050	Feature coordinate system (command system) (D)	Feature coordinate system (command system) (D)	Feature coordinate system (command system) (D)	Workpiece coordinate system (command system) (C)
#151051 - #151100	Workpiece coordinate system (command system) (C)	Workpiece coordinate system (command system) (C)	Workpiece coordinate system (machine system) (B)	Workpiece coordinate system (machine system) (B)
#151101 - #151150	Machine coordinate system (A)			

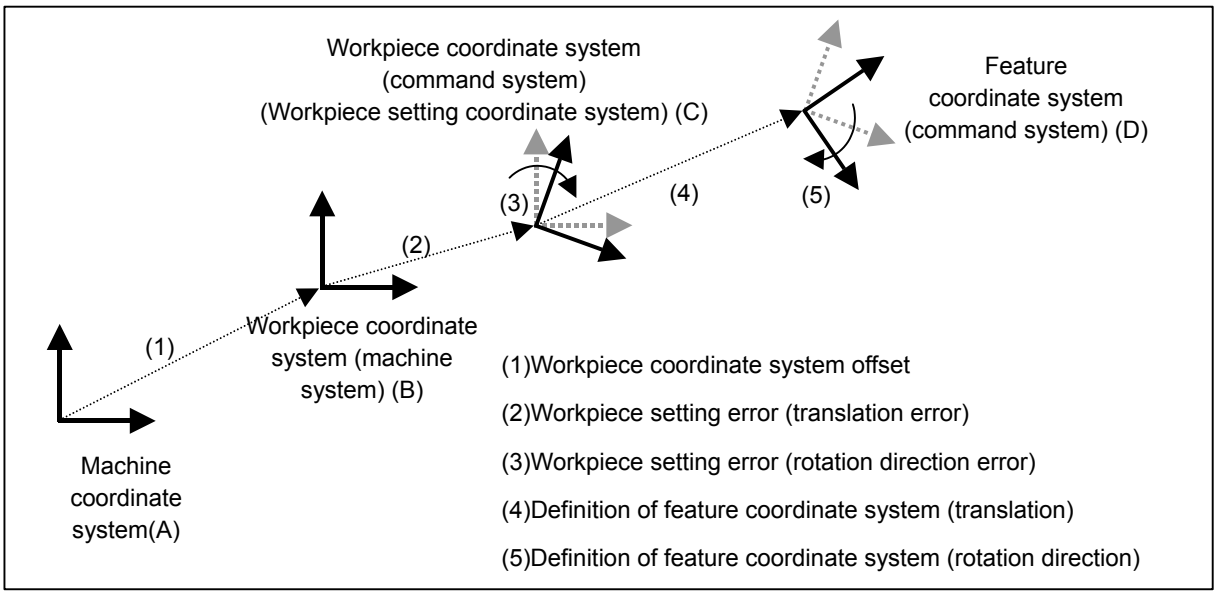


Fig.15.5 (w) Coordinate system that can be read skip positions

This example uses following settings.

Bit 5 (LV3) of parameter No. 5400 is set to 0.
Bit 3 (WSK) of parameter No. 11200 is set to 0.
Parameter No.19680 is set to 2. (Tool rotation type)
Parameter No.19666 is set to 0. (Tool holder offset value)

Parameters Nos.19709 to 19714 are set to 0. (Intersection offset vector)

Workpiece setting error compensation axes
X, Y, Z, B, C

G54 Workpiece coordinate system offset
X0.0
Y0.0
Z-150.0
B0.0
C0.0

Workpiece setting error compensation value
x0.0
y0.0
z0.0
a0.0
b.0
c45.0

Sample program
```
O0001
N100G90G00X0Y0Z0B0C0
N200G54
N300G5.1Q1
N400G54.4P1
N500X0Y0Z0B0C0
N600G68.2X0Y0Z0I45.0J0K0
N700G53.1
N800G31X100.0F10.0
      :
```

If sample program O0001 is executed, tool center point moves +X axis direction on feature coordinate system (+Y axis direction on machine coordinate system) at N800 block.
Therefore, if skip signal is inputted at N800 block, skip position becomes the position as Table15.5 (m).

Table15.5 (m) Skip positions of each system variables

System variables	Skip positions				
	X axis	Y axis	Z axis	B axis	C axis
#100151 -	0.000	0 and over	0.000	0.000	45.000
#151001 -	0 and over	0.000	0.000	0.000	0.000
#151051 -	0 and over	0 and over	0.000	0.000	0.000
#151101 -	0.000	0 and over	-150.000	0.000	45.000

- Rapid traverse in a canned cycle for drilling

Using bit 1 (D3R) of parameter No. 11221, it is possible to select the rapid traverse mode in the drilling direction in any of the tilted working plane indexing mode, 3-dimensional coordinate conversion mode, and workpiece setting error compensation mode. When bit 1 (D3R) of parameter No. 11221 is set to 0, the cutting feed mode is selected; when it is set to 1, the rapid traverse mode is selected. For positioning to the initial level at the start of a canned cycle for drilling, however, the rapid traverse mode is used, regardless of whether bit 1 (D3R) of parameter No. 11221 is set to 0 or 1. Rapid traverse override can be applied in the rapid traverse mode, and cutting feed override can be applied in the cutting feed mode. Bit 1 (D3R) of parameter No. 11221 can also be used to change the mode of rapid traverse in the rigid tapping direction.

For acceleration/deceleration after interpolation

When bit 1 (D3R) of parameter No. 11221 is set to 1 (for the rapid traverse mode), rapid traverse in the drilling direction in a canned cycle for drilling in the tilted working plane indexing mode, 3-dimensional coordinate conversion mode, or workpiece setting error compensation mode is accelerated/decelerated along each axis independently according to the time constants specified in parameters Nos. 1620 and 1621. So, when performing positioning of linear interpolation type with D3R is set to 1, make the following settings:

Bit 1 (LRP) of parameter No. 1401=1	Selects positioning of linear interpolation type.
Bit 4 (PRT) of parameter No. 1603=1	Selects acceleration/deceleration of constant time type.
Parameter No.1620	Set the same value for all.
Parameter No.1621	Set the same value for all.

For acceleration/deceleration before interpolation

In a canned cycle for drilling, linear rapid traverse in the drilling direction is performed at all times.

Feedrate

When bit 1 (D3R) of parameter No. 11221 is set to 1 (for the rapid traverse mode), the rapid traverse rate in the drilling direction in a canned cycle for drilling in the tilted working plane indexing mode, 3-dimensional coordinate conversion mode, or workpiece setting error compensation mode is clamped to a parameter value indicated below if the rapid traverse override is 100%. In the rapid traverse override F0 mode, however, the rapid traverse rate is clamped to the value of parameter No. 1421 or a parameter value listed in the Table 15.5 (n), “Feedrate in the rapid traverse mode”, whichever smaller.

Table 15.5 (n) Feedrate in the rapid traverse mode

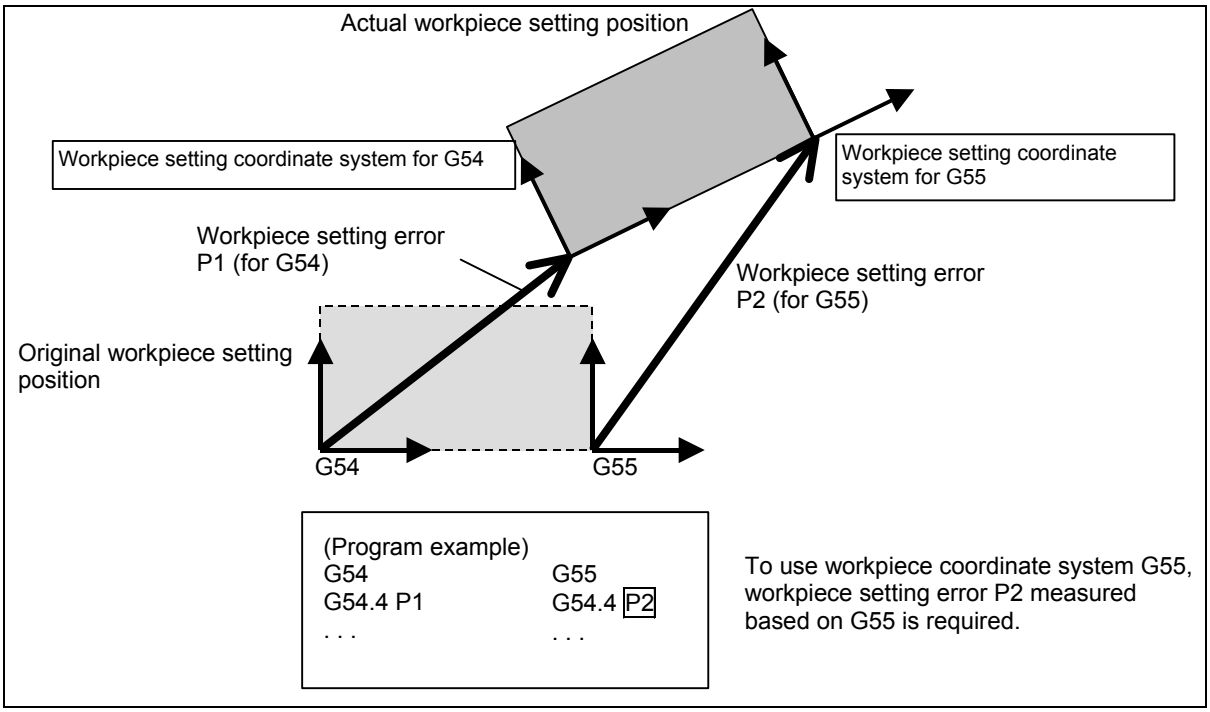
When the parameter No.5412 is set to 0	Parameter No.1420
When the parameter No.5412 is not set to 0	Value of parameter No. 1420 or value of parameter No. 5412, whichever smaller

However, when the external deceleration function is enabled, the feedrate is clamped to a parameter value listed above that is overridden or an external deceleration speed, whichever smaller.

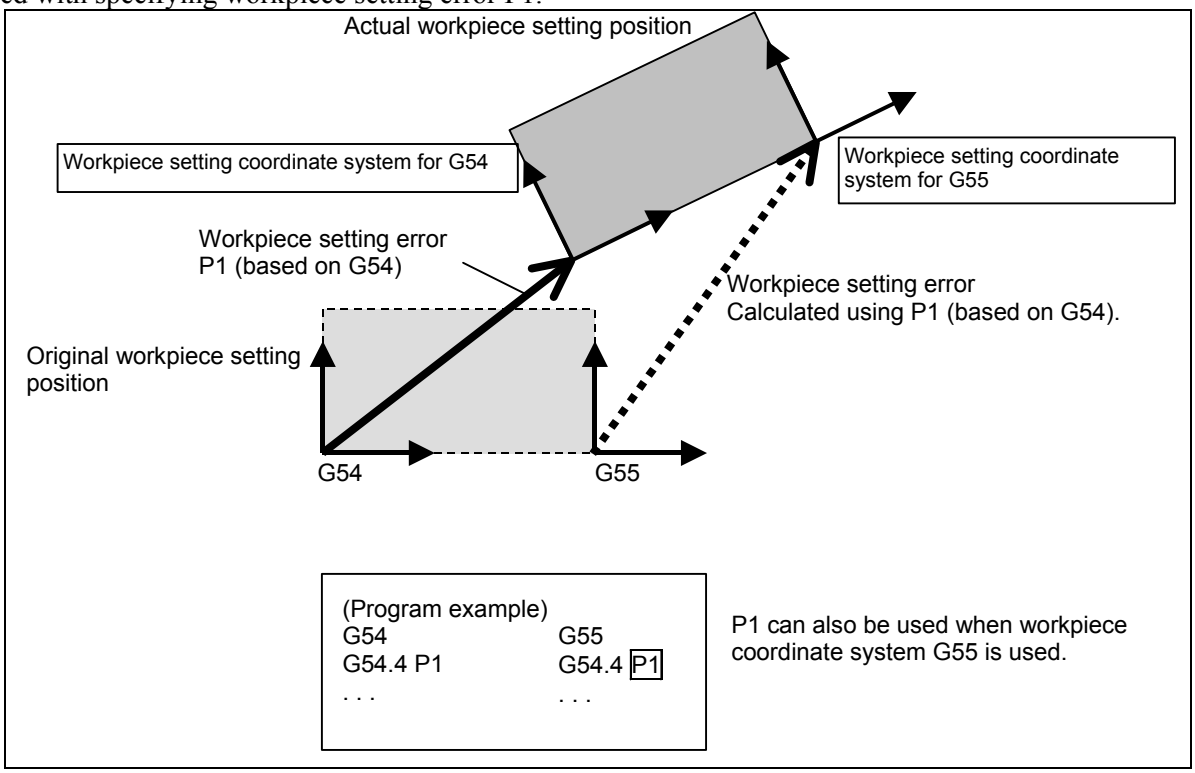
NOTE
1 Set a rapid traverse rate for each axis in parameter No. 1420 and a tangential rapid traverse rate in parameter No. 5412.
2 If rapid traverse of nonlinear interpolation type is performed, the feedrate is clamped to the value of parameter No. 1420, regardless of the setting of parameter No. 5412.
3 When the cutting feed mode is selected (when bit 1 (D3R) of parameter No. 11221 is set to 0), set a tangential rapid traverse rate in parameter No. 5412.

To use one workpiece setting error for multiple workpiece coordinate systems

Workpiece setting errors Δx, Δy, and Δz are measured based on a certain workpiece coordinate system. When parameters Nos. 11411 to 11417 are set to 0, to use multiple workpiece coordinate systems for a workpiece setting state, as many sets of workpiece setting errors Δx, Δy, and Δz as the number of workpiece coordinate systems to be used must be set.



In this case, the workpiece coordinate system on which workpiece setting errors are based can be set using parameters Nos. 11411 to 11417 to use one workpiece setting error for multiple workpiece coordinate system. With this function, setting workpiece setting errors and the reference workpiece coordinate system in a certain workpiece setting state using parameters Nos. 11411 to 11417 allows the workpiece setting errors to be used for another workpiece coordinate system.
In the example below, setting the reference workpiece coordinate system for workpiece setting error P1 to G54 (setting parameter No. 11411 to 54) allows the workpiece setting coordinate system set in G55 to be used with specifying workpiece setting error P1.



The workpiece offset value for each rotation axis is not considered. Set the same workpiece offset value for each rotation axis as in the reference workpiece coordinate system.

Set the number of the workpiece coordinate system on which each of workpiece setting errors Nos. 01 to 07 is based in parameters Nos. 11411 to 11417. For G54 to G59, set 54 to 59. For G54.1P1 to G54.1P300, set 1001 to 1300.
When one of the parameters is set to 0, the relevant workpiece setting error cannot be used in multiple workpiece coordinate systems.
If an invalid workpiece coordinate system number is set, alarm PS0436, "ILLEGAL PARAMETER IN WSC" is issued in the block in which workpiece setting error compensation is started.

Examples

- Example 1

O1 represents a program that cuts each side of a square.

```
O1 ;
N10 G55 ;                          Set coordinate system
N20 G90 G00 X0 Y0 Z300.0 B0 C0 ;   Move to initial position
N30 G01 G43 H01 Z40.0 F500. ;      Start tool length compensation
                                       H01 is tool length compensation number.
N40 X50.0 Y50.0 Z20.0 ;            Z-axis height of machining plane is 20.0.
N50 X150.0 ;
N60 Y150.0 ;
N70 X50.0 ;
N80 Y50.0 ;
N90 X0 Y0 Z40.0 ;
N100 G49 Z300.0 ;                  Cancel tool length compensation
                                       Move to initial position on Z-axis
N110 M30;
```

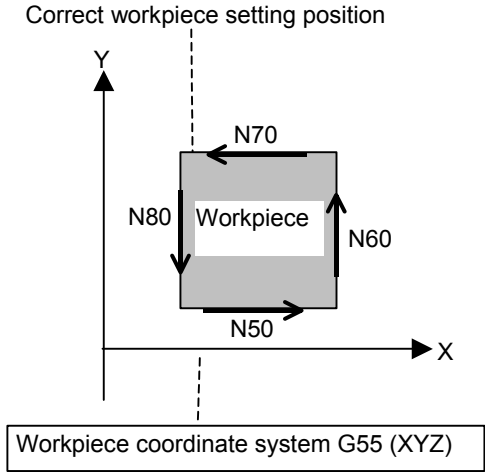


Fig. 15.5 (x) Operation when there is no workpiece setting error

Suppose that the workpiece is displaced from the "correct workpiece setting position" as shown in Fig. 15.5 (y).

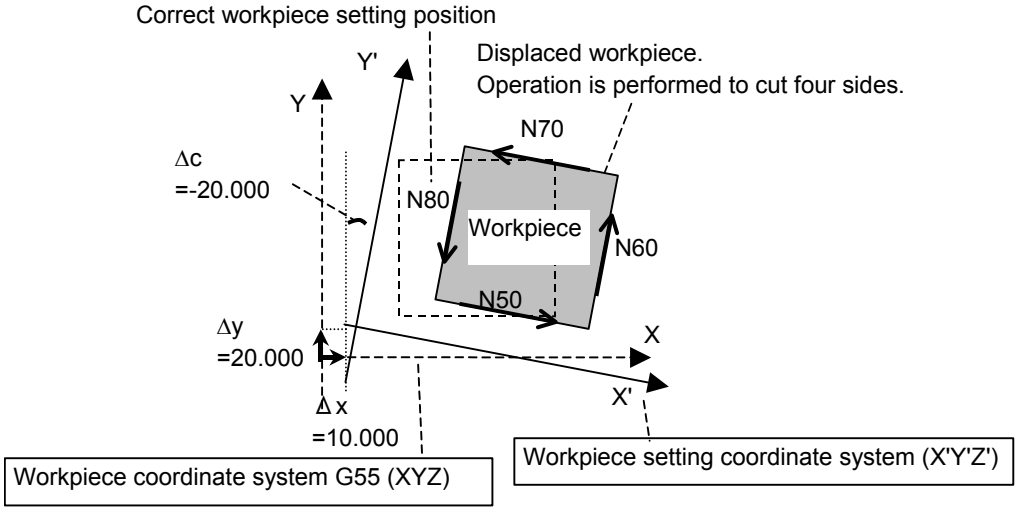


Fig. 15.5 (y) Operation when there is a workpiece setting error

The workpiece coordinate system, when rotated by -20.000 deg about the Z-axis, shifted by 10.000 in the X direction, and shifted by 20.000 in the Y direction, is to match the workpiece setting coordinate system. At this time, set the following workpiece setting error values:

(Workpiece setting error No. 01)
x 10.000
y 20.000
z 0.000
a 0.000
b 0.000
c -20.000

To validate the workpiece setting error, add N15 and N115 to O1 as indicated below.

```
O1 ;
N10 G55 ;                          Set coordinate system
N15 G54.4 P1                       Workpiece setting error compensation mode ON
N20 G90 G00 X0 Y0 Z300.0 B0 C0 ;   Move to initial position
N30 G01 G43 H01 Z40.0 F500. ;      Start tool length compensation
                                       H01 is tool length compensation number.
N40 X50.0 Y50.0 Z20.0 ;            Z-axis height of machining plane is 20.0.
N50 X150.0 ;
N60 Y150.0 ;
N70 X50.0 ;
N80 Y50.0 ;
N90 X0 Y0 Z40.0 ;
N100 G49 Z300.0 ;                  Cancel tool length compensation
                                       Move to initial position on Z-axis
N115 G54.4 P0 ;                    Workpiece setting error compensation mode OFF
N110 M30;
```

When O1 is executed, the tool moves to cut each side of the displaced workpiece as shown by the solid lines in Fig. 15.5 (y).

- Example 2

O2 is a program for cutting each side of a square by using tool center point control.

The machine is of tool rotation type, the C-axis is the master rotation axis and rotates about the Z-axis, and the B-axis is the slave axis and rotates about the Y-axis. For cutting on the plane normal to the movement direction, the tool is tilted 45 deg relative to the +Z direction.

```
O2 ;
N10 G55 ;                              Set coordinate system
N20 G90 G00 X0 Y0 Z300.0 B0 C0 ;       Move to initial position
N30 G01 G43.4 H01 Z40.0 F500. ;        Start tool center point control
                                           H01 is tool length compensation number.
N40 X50.0 Y50.0 Z20.0 B45.0 C-90.0 ;   Z-axis height of machining plane is 20.0.
N50 X150.0 ;
N60 C0.0 ;
N70 Y150.0 ;
N80 C90.0 ;
N90 X50.0 ;
N100 C180.0 ;
N110 Y50.0 ;
N120 X0 Y0 Z40.0 B0.0 C0.0 ;
N130 G49 Z300.0 ;                      Cancel tool length compensation
                                           Move to initial position on Z-axis
N140 M30;
```

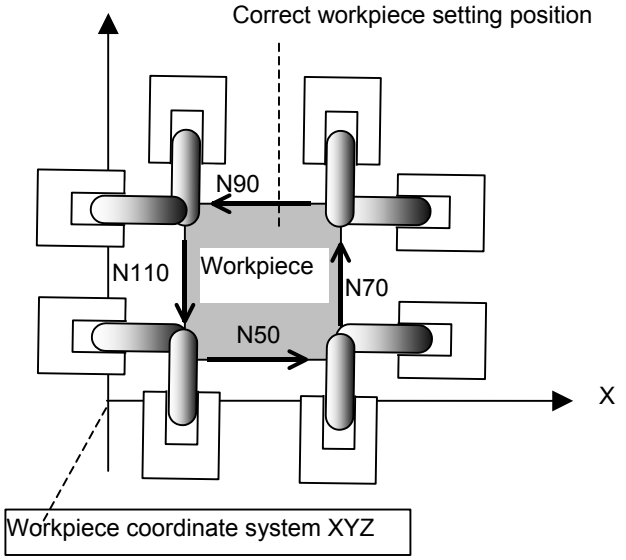


Fig. 15.5 (z) Operation of tool center point control when there is no workpiece setting error

Suppose that the workpiece is displaced from the correct workpiece setting position as with example 1, and set a workpiece setting error in the same was as in Example 1.
Moreover, add N15,N16,N135 and N136 to O2 as indicated below to validate the workpiece setting error as in the case of Example 1.

```
O2 ;
N10 G55 ;                              Set coordinate system
N15 G05.1 Q1                           AI contour control mode ON
N16 G54.4 P1                           Workpiece setting error compensation mode ON
N20 G90 G00 X0 Y0 Z300.0 B0 C0 ;       Move to initial position
N30 G01 G43.4 H01 Z40.0 F500. ;        Start tool center point control
                                           H01 is tool length compensation number.
N40 X50.0 Y50.0 Z20.0 B45.0 C-90.0 ;   Z-axis height of machining plane is 20.0.
N50 X150.0 ;
N60 C0.0 ;
N70 Y150.0 ;
N80 C90.0 ;
N90 X50.0 ;
N100 C180.0 ;
N110 Y50.0 ;
N120 X0 Y0 Z40.0 B0.0 C0.0 ;
N130 G49 Z300.0 ;                      Cancel tool length compensation
                                           Move to initial position on Z-axis
N135 G54.4 P0 ;                        Workpiece setting error compensation mode OFF
N136 G05.1 Q0 ;                        AI contour control mode OFF
N140 M30;
```

Set bit 0 (RCM) of parameter No. 11200 to 1 to perform tool direction compensation, then execute O2. When the tool moves to cut each side of the workpiece with the tool tip, the tool is tilted 45 deg relative to the +Z direction on the plane normal to the movement direction as shown in Fig. 15.5 (aa).

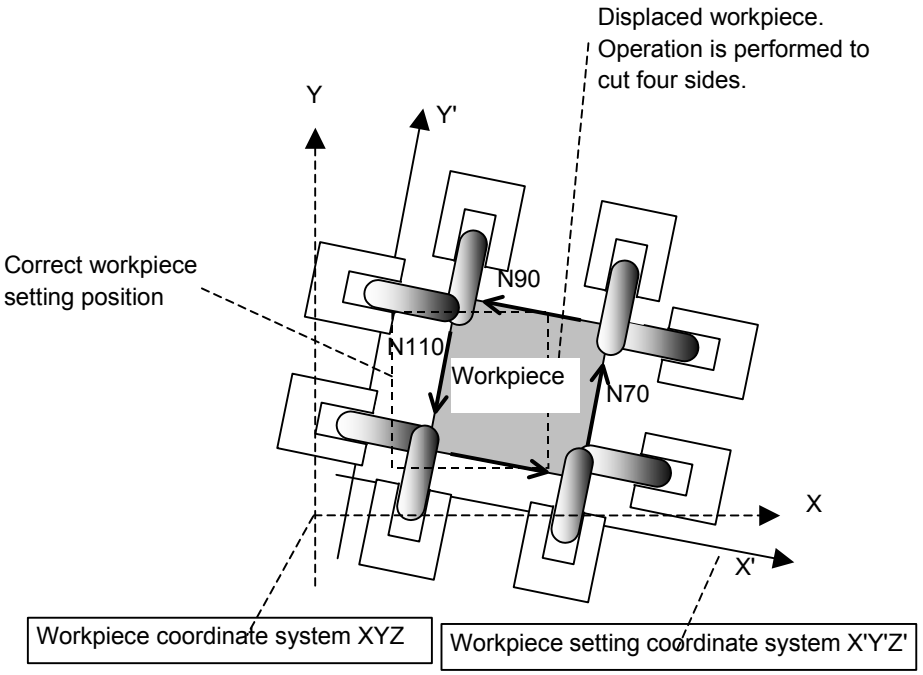


Fig. 15.5 (aa) Operation of tool center point control when there is a workpiece setting error

Restrictions

Described below is the restriction related to workpiece setting compensation errors.

- Specifiable G codes

When workpiece setting error compensation is enabled, the G codes listed below can be specified.

G00	Positioning
G01	Linear interpolation

G02	Circular interpolation (CW)
G03	Circular interpolation (CCW)
G04	Dwell
G05.1 Q0/Q1	AI contour control mode OFF/ON
G10	Programmable data input
G11	Programmable data input mode cancel
G17	Plane selection (XY)
G18	Plane selection (ZX)
G19	Plane selection (YZ)
G28	Reference position return
G29	Return from reference position
G30	2nd/3rd/4th reference position return
G31	Skip function
G40	Tool radius · tool nose radius compensation / 3-dimensional cutter compensation cancel
G41	Tool radius/tool nose radius compensation left / 3-dimensional cutter compensation left
G41.2/G41.4/G41.5	3-dimensional cutter compensation left (type 1) (only if also tool center point control is used)
G41.6	3-dimensional cutter compensation left (type 2) (only if also tool center point control is used)
G42	Tool radius/tool nose radius compensation right / 3-dimensional cutter compensation right
G42.2/G42.4/G42.5	3-dimensional cutter compensation right (type 1) (only if also tool center point control is used)
G42.6	3-dimensional cutter compensation right (type 2) (only if also tool center point control is used)
G43	Tool length compensation +
G43.4	Tool center point control (type 1)
G43.5	Tool center point control (type 2)
G49 (G49.1)	Tool length compensation cancel
G50	Scaling cancel
G51	Scaling
G50.1	Programmable mirror image cancel
G51.1	Programmable mirror image
G53	Machine coordinate system selection
G53.1	Tool axis direction control
G65	Custom macro call
G66	Custom macro modal call
G67	Custom macro modal call cancel
G68.2	Tilted working plane indexing
G73, G74, G76, G80 to G89	Canned cycle for drilling

M

G69	Coordinate system rotation/3-dimensional coordinate system conversion/Tilted working plane indexing cancel
G90	Absolute programming
G91	Incremental programming
G94	Feed per minute
G95	Feed per revolution
G98	Canned cycle : return to initial level
G99	Canned cycle : return to R point level

T

G69.1 Coordinate system rotation / 3-dimensional coordinate system conversion/Tilted working plane indexing cancel
G90 Absolute programming (for G code system B and C)
G91 Incremental programming (for G code system B and C)
G94 Feed per minute (for G code system B and C)
G95 Feed per revolution (for G code system B and C)
G98 Canned cycle : return to initial level (for G code system B or C)
G99 Canned cycle : return to R point level (for G code system B or C)

- Modal G codes usable when workpiece setting error compensation is specified

In a modal G code state listed below, workpiece setting error compensation can be specified.

G00	Positioning
G01	Linear interpolation
G13.1	Polar coordinate interpolation mode cancel
G15	Polar coordinate command cancel
G17	Plane selection (XY)
G18	Plane selection (ZX)
G19	Plane selection (YZ)
G20	Inch input
G21	Metric input
G23	Stored stroke check function OFF
G25	Spindle speed fluctuation detection OFF
G40	Tool radius · tool nose radius compensation / 3-dimensional cutter compensation cancel
G49 (G49.1)	Tool length compensation cancel
G50	Scaling cancel
G50.1	Programmable mirror image cancel
G50.2	Polygon turning cancel
G54 to G59, G54.1	Workpiece coordinate system selection
G64	Cutting mode
G67	Custom macro modal call cancel
G80	Canned cycle cancel
G80.5 (G80.8)	Electronic gear box synchronization cancel
G97	Constant surface speed control cancel

M

G40.1 Normal direction control cancel
G54.2 P0 Rotary table dynamic fixture offset cancel
G69 Coordinate system rotation/3-dimensional coordinate system conversion/Tilted working plane indexing cancel
G90 Absolute programming
G91 Incremental programming
G94 Feed per minute
G95 Feed per revolution
G98 Canned cycle initial level return
G99 Canned cycle R point level return

T

G69 Mirror image for double turret OFF/balanced cutting mode cancel

G69.1	Coordinate system rotation/3-dimensional coordinate system conversion/Tilted working plane indexing cancel
G90	Absolute programming (for G code system B and C)
G91	Incremental programming (for G code system B and C)
G94	Feed per minute (for G code system B and C)
G95	Feed per revolution (for G code system B and C)
G98	Canned cycle initial level return (for G code system B and C)
G99	Canned cycle R point level return (for G code system B and C)

- Manual intervention and handle interruption

In the workpiece setting error compensation mode, do not perform manual intervention and handle interruption.

- Rapid traverse command

When using workpiece setting error compensation, specify linear rapid traverse (by setting set bit 1 (LRP) of parameter No. 1401 to 1).

- Relationships with other modal commands

The commands for the functions listed below must be nested with a workpiece setting error compensation command and must be placed between G54.4 P_ and G54.4 P0.
Namely, specify G54.4 P_ when the mode of a desired function below is off. Next, turn on then off the mode of the function, then specify G54.4 P0.

- Cutter compensation (G40, G41, G41.2, G41.3, G41.4, G41.5, G41.6, G42, G42.2, G42.4, G42.5, G42.6)
- Tool length compensation (G43, G43.4, G43.5, G49)
- Programmable mirror image (G50.1, G51.1)
- Scaling (G51)
- Tilted working plane indexing (G68.2, G69)
- Canned cycle (G73,G74,G76,G80 to G89)

- PMC axis control

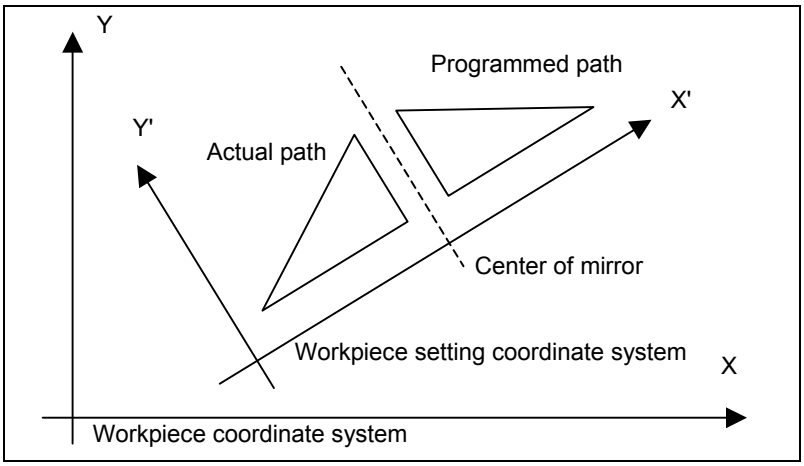
In the workpiece setting error compensation mode, do not perform PMC axis control to the axes which are for workpiece setting error compensation.

- Movement of start-up and cancel of Tool length offset

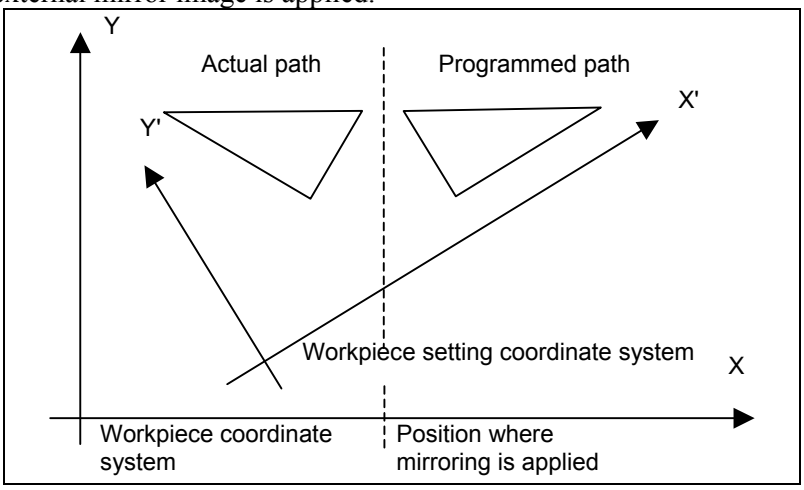
To perform tool length compensation (including tool center point control) in the workpiece setting error compensation mode, set bit 6 (TOS) of parameter No. 5006 or bit 2 (TOP) of parameter No. 11400 to 1 (perform tool length compensation by shifting a coordinate system). If parameters TOS and TOP are 0, alarm PS0438, ”ILLEGAL PARAMETER IN TOOL DIRC CMP” is issued.

- Mirror image

If an attempt is made to use workpiece setting error compensation and programmable mirror image simultaneously, programmable mirror image is applied to coordinates in the workpiece setting coordinate system, then workpiece setting error compensation is performed.



If an attempt is made to use workpiece setting error compensation and external mirror image (using the mirror image signal or mirror image setting) simultaneously, workpiece setting error compensation is applied first, then external mirror image is applied.



- Stroke limit check before movement

In the mode of Workpiece setting error compensation, Stroke limit check before movement is not available.

- In the case that Tool posture control is used with Type-2 of Tool center point control

In the case that Tool posture control is used with Type-2 of Tool center point control, when the angle between a tool posture and the singular posture is less than the parameter No.19738, the tool posture at the end of the block is changed so that the tool posture passes the singular posture.
But, when tool direction is compensated (bit 0 (RCM) of parameter No.11200 = 1), the tool posture may not pass the singular posture because of the compensation after tool posture control.

- Program restart

Do not restart a program if workpiece setting error compensation is being performed for it.

- Reset

Workpiece setting error compensation mode is canceled by a reset.

Performing 3-dimensional coordinate system conversion or tilted working plane indexing in the workpiece setting error compensation mode causes the 3-dimensional coordinate system conversion mode and tilted working plane indexing mode to be canceled by a reset no matter what the setting of Bit 2 (D3R) of parameter No. 5400.

Performing tool length compensation (including tool center point control) in the workpiece setting error compensation mode causes the tool length compensation mode and tool length compensation vector to be canceled by a reset no matter what the setting of the following parameters:

- Bit 6 (CLR) of parameter No.3402
- Bit 0 (C08) of parameter No.3407
- Bit 6 (LVK) of parameter No.5003

- Synchronous slave axis absolute coordinates

If feed axis synchronization is in effect, no slave axis absolute coordinate is displayed properly.

- Workpiece coordinate system shift

Do not perform workpiece setting error compensation with G52 (local coordinate system setting) or G92/G50 (T series) (workpiece coordinate system shift) executed.
Once the workpiece coordinate system has been shifted, preset it using G92.1 or axis-specific workpiece coordinate system preset functions before starting workpiece setting error compensation.

- Manual reference position return

If manual reference position return is performed in the workpiece setting error compensation mode, alarm PS5324, “REFERENCE RETURN INCOMPLETE” is issued. If you want to perform manual reference position return, cancel the workpiece setting error compensation mode in advance.

15.6 TOOL OFFSET FOR MILLING AND TURNING FUNCTION (G43.7)

Overview

In the past, the tool offset screen for the machining center system was different from that for the lathe system. This function enables display and manipulation of offset data on a common tool offset screen.
By enabling this function, it is possible to make offset data into a configuration suitable to complex machining, so that tool offset for the lathe system can also be performed in the machining center system.

Format

M

```
G43.7 Hα;      Tool offset mode ON
      :
      :
G49; or H0;    Tool offset cancel

  α: Specify an H code, followed by an offset number.
```

T

Conforms to the format for the lathe system.

Explanation

- Tool offset (G43.7)

M

By specifying a G43.7 command with an H code, it is possible to perform tool offset. This function performs tool offset by using the offset amounts for the X-, Y-, and Z-axes as part of the offset data for tool offset specified with an H code.

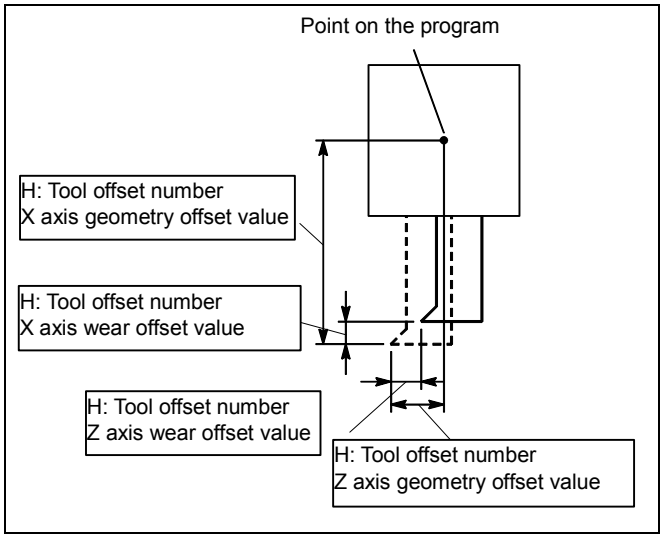


Fig. 15.6 (a) Correspondence between a specified H code and tool geometry offset and tool wear offset (example with the X- and Z-axes)

Offset with tool movement and offset with coordinate shift

M

The sum of geometry offset and wear offset is performed as the type specified with bit 6 (TOS) of parameter No. 5006 or bit 2 (TOP) of parameter No. 11400. It is not possible to set the types of geometry offset and wear offset separately.

- Relationships between tool length compensation and tool offset

M

If the G43.7 command is executed in G43 mode, tool length compensation, which has been enabled in G43 mode, is automatically canceled, and the tool offset of the G43.7 command is enabled instead.
If the G43 command is executed in G43.7 mode, tool offset, which has been enabled in G43.7 mode, is automatically canceled, and the tool length compensation of the G43 command is enabled instead.

- Tool length compensation on the Z-axis

Of the compensation amounts used with the functions below, only the value set for "Z/LENGTH" on the tool offset screen are used. Values set for "X", "Y", "NOSE R/RAD", etc. are not used.

G43 / G44: Tool length offset
G43.1: Tool length compensation in tool axis direction
G43.4: Tool center point control (type 1)
G43.5: Tool center point control (type 2)

- Parameters related to tool offset

M

The following are parameters that are effective even in G43.7 mode and those that are not.

Parameters that are effective even in G43.7 mode

Bit 6 (LVK) of parameter No. 5003
Bit 6 (TOS) of parameter No. 5006
Bit 6 (3OC) of parameter No. 5007
Bit 7 (3OF) of parameter No. 5007
Bit 2 (TOP) of parameter No. 11400

Parameters that are not effective in G43.7 mode
Bits 0 (TLC) and 1 (TLB) of parameter No. 5001
Regardless of these settings, tool offset is effective to the X- and Y-axes.
Bit 3 (TAL) of parameter No. 5001
Regardless of this setting, the operation of 1, "Not generate an alarm even if two or more axes are offset", is performed.
Bit 6 (EVO) of parameter No. 5001
Regardless of this setting, the operation of 0, "The new value becomes valid in a block where G43, G44, G43.7, or an H code is specified next." is performed.

Automatic tool length measurement (M series)

M

If the tool offset function for complex machining is used, the value measured in automatic tool length measurement (M series) G37 is input as the tool length (Z/length). By setting bit 4 (MMT) of parameter No. 11403, it is possible to input it as the axis subject to tool offset (X, Y, Z/length).
Example)
If measurement is performed on the X-axis, the value is input as the offset value for the X-axis.
If measurement is performed on the Y-axis, the value is input as the offset value for the Y-axis.
If measurement is performed on the Z-axis, the value is input as the offset value for the Z-axis.

- Direct input of offset value measured B

M

If the tool offset function for complex machining is used, the value measured in direct input of offset value measured B is input as the tool length (Z/length). By setting bit 4 (MMT) of parameter No. 11403, it is possible to input it as the axis subject to tool offset (X, Z/length).
Example)
If measurement is performed on the X-axis, the value is input as the offset value for the X-axis.
If measurement is performed on the Z-axis, the value is input as the offset value for the Z-axis.

- Tool length/workpiece zero point measurement

M

The value measured in tool length/workpiece zero point measurement is input as the tool length (Z/length).

Programmable data input (G10)

The data format used to input/output offset data on the tool offset screen and to input offset data due to program operation is the G10 format, below.

- G10 data format

G10 Lbaaa ＋ Various tool offset formats

- L format: Lbaaa

  aaa : 200- Tool offset, tool-nose radius compensation (cutter compensation), virtual tool tip direction
  Tool wear compensation
  : 201- Tool offset, tool-nose radius compensation (cutter compensation), virtual tool tip direction
  Tool geometry compensation
  : 202- Corner R offset in the cutting point command
  Tool wear compensation
  : 203- Corner R offset in the cutting point command
  Tool geometry compensation

  b : 0- Writes the specified data over the existing data.
  : 1- Adds the specified data to the existing data.

- Various tool offset formats

  aaa : 200,201- Tool offset, tool-nose radius compensation (cutter compensation), virtual tool tip direction
  P_ : Tool offset number
  X_ : Tool offset amount on the X-axis
  Z_ : Tool offset amount on the Z-axis (tool length compensation)
  R_ : Tool-nose radius compensation amount (cutter compensation amount)
  Q_ : Virtual tool tip direction
  Y_ : Tool offset amount on the YX-axis
  : 202,203- Corner R offset in the cutting point command
  P_ : Tool offset number
  R_ : Corner R offset

- Example

```
O0010;
G10L200P1X_Z_R_Q_Y_;   Overwrite tool wear compensation, such as tool offset.
G10L1200P1X_Z_R_Q_Y_;  Add tool wear compensation, such as tool offset.
G10L201P1X_Z_R_Y_;     Overwrite tool geometry compensation, such as tool offset.
G10L1201P1X_Z_R_Y_;    Add tool geometry compensation, such as tool offset.
:
:
G10L202P1R_;           Overwrite corner R offset and tool wear compensation.
G10L1202P1R_;          Add corner R offset and tool wear compensation.
G10L203P1R_;           Overwrite corner R offset and tool geometry compensation.
G10L1203P1R_;          Add corner R offset and tool geometry compensation.
M02;
```

- Input in the G10 format of the machining center system

M

Offset data can be input in the G10 format of the machining center system. Note, however, that only the offset data specified in the L format is input.

```
G10G90L10P1R1.001
G10G90L11P1R2.001
G10G90L12P1R3.001
G10G90L13P1R4.001
```

G10G90L110P1R5.001
G10G90L111P1R6.001

The offset data that supports the L format is:

L10 : Z axis tool offset (geometry)
L11 : Z axis tool offset (wear)
L12 : Tool offset (geometry)
L13 : Tool offset (wear)
L110 : Corner R offset (geometry)
L111 : Corner R offset (wear)

NOTE
Input in the G10 format of the machining center system is available only if the machining center system is selected with parameter No. 983 (1 is set).

- Input in the G10 format of the lathe system

T

Offset data can be input in the G10 format of the lathe system.
G10P1X1.001Z2.001R3.001Q4Y5.001

Pn : Tool offset number (1 to number of tool compensation values)
Pn ≤ 10000 : Tool wear compensation
Pn ≥ 10001 : Tool geometry compensation
X_ : X axis tool offset
Z_ : Z axis tool offset
R_ : Tool nose radius compensation
Q_ : Virtual tool tip direction
Y_ : Y axis tool offset

NOTE
Input in the G10 format of the lathe system is available only if the lathe system is selected with parameter No. 983 (0 is set).

Custom macro system variables

M

When bit 3 (V15) of parameter No. 6000 is set to 0

System variable number	System variable name	Attribute	Description
#2001 to #2200	[#_OFSHW[n]] or [#_OFSZW[n]]	R/W	Z axis tool offset (wear) (Old name: Tool compensation value (H code, wear)) Note) Subscript n represents a compensation number (1 to 200).
#10001 to #10999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2201 to #2400	[#_OFSHG[n]] or [#_OFSZG[n]]	R/W	Z axis tool offset (geometry) (Old name: Tool compensation value (H code, geometry)) Note) Subscript n represents a compensation number (1 to 200).
#11001 to #11999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#12001 to #12999	[#_OFSDW[n]] or [#_OFSRW[n]]	R/W	Tool offset (wear) (Old name: Tool compensation value (D code, wear)) Note) Subscript n represents a compensation number (1 to 999).
#13001 to #13999	[#_OFSDG[n]] or [#_OFSRG[n]]	R/W	Tool offset (geometry) (Old name: Tool compensation value (H code, geometry)) Note) Subscript n represents a compensation number (1 to 999).

When bit 3 (V15) of parameter No. 6000 is set to 1

System variable number	System variable name	Attribute	Description
#2001 to #2200	[#_OFSHG[n]] or [#_OFSZG[n]]	R/W	Z axis tool offset (geometry) (Old name: Tool compensation value (H code, geometry)) Note) Subscript n represents a compensation number (1 to 200).
#10001 to #10999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2201 to #2400	[#_OFSHW[n]] or [#_OFSZW[n]]	R/W	Z axis tool offset (wear) (Old name: Tool compensation value (H code, wear)) Note) Subscript n represents a compensation number (1 to 200).
#11001 to #11999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2401 to #2600	[#_OFSDG[n]] or [#_OFSRG[n]]	R/W	Tool offset (geometry) (Note 1) (Old name: Tool compensation value (H code, geometry)) Subscript n represents a compensation number (1 to 200). Note 1) Effective if bit 5 (D15) of parameter No. 6004 is 1.
#12001 to #12999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2601 to #2800	[#_OFSDW[n]] or [#_OFSRW[n]]	R/W	Tool offset (wear) (Note 1) (Old name: Tool compensation value (D code, wear)) Subscript n represents a compensation number (1 to 200). Note 1) Effective if bit 5 (D15) of parameter No. 6004 is 1.
#13001 to #13999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).

System variables not dependent on bit 3 (V15) of parameter No. 6000

System variable number	System variable name	Attribute	Description
#21001 to #21999	[#_CORR_G[n]]	R/W	Corner R offset (geometry) Note) Subscript n represents a compensation number (1 to 999).
#22001 to #22999	[#_CORR_W[n]]	R/W	Corner R offset (wear) Note) Subscript n represents a compensation number (1 to 999).
#23001 to #23999	[#_OFST[n]]	R/W	Virtual tool tip T position Note) Subscript n represents a compensation number (1 to 999).
#135001 to #135999	[#_OFSXW[n]]	R/W	X axis tool offset (wear) Note) Subscript n represents a compensation number (1 to 999).
#136001 to #136999	[#_OFSYW[n]]	R/W	Y axis tool offset (wear) Note) Subscript n represents a compensation number (1 to 999).
#137001 to #137999	[#_OFSXG[n]]	R/W	X axis tool offset (geometry) Note) Subscript n represents a compensation number (1 to 999).
#138001 to #138999	[#_OFSYG[n]]	R/W	Y axis tool offset (geometry) Note) Subscript n represents a compensation number (1 to 999).

T

Conforms to the conventional custom macro system variables.

Limitation

(1) Functions that cannot be used together with this function
- Tool offset B function
- Changing Active Offset Value with Manual Move
- Basic Operation Package 2 function
- MANUAL GUIDE *i*

If any of the above functions is ordered together with this function, the tool offset screen is replaced by the conventional screen, so that G43.7 cannot be used.

(2) If this function is used, part of the specifications of the function below is limited.
Tool offset protection cannot be performed with the 8-level data protection function.

Note

NOTE
This function is an optional function.

15.7 TOOL OFFSET CONVERSION FUNCTION (G44.1)

Overview

In the complicated machine composition that has rotation axes, the tool offset of each axis and the direction of imaginary tool nose are changed by angle of rotation degree of the tool nose rotation axis and the swivel head axis.
In this function, the tool offset and the direction of imaginary tool nose (which are set to the tool offset screen) are converted into appropriate value that depends on angle of the tool by G44.1 command.

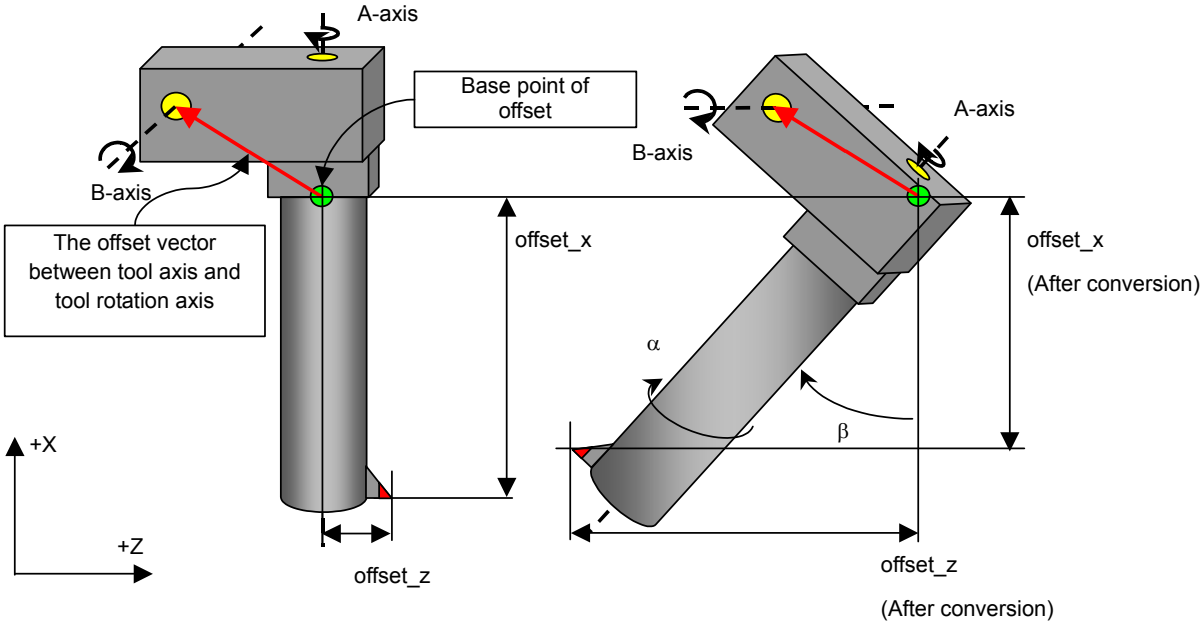


Fig.15.7 (a) Change of the tool offset by angle of A-axis,B-axis

In this specification, it is assumed that the tool nose rotation axis is A-axis rounding on X-axis and the swivel head axis is B-axis rounding on Y-axis in X-Z plane. The commanded angle of A-axis must be selected between 0 and 180 degree.

NOTE
Using this function requires the option “Tool offset conversion function” and “Tool offset for Milling and Turning function”

T

NOTE
Using this function requires the setting of “Extended tool selection function”

Format

The program format is shown below.

T

G44.1 Dα Pn; Tool offset mode ON
:
:
G49; or D0; Tool offset cancel

Dα: Offset number is commanded with D-code.
If D-code is not commanded in G44.1 block, alarm PS0536, "ILLEGAL COMMAND IN G44.1" occurs.
Pn: The direction of imaginary tool nose is commanded.
If P is not commanded, the direction of imaginary tool nose is selected according to the angle of the tool nose rotation axis (A-axis) and the swivel head axis (B-axis) automatically.

M

G44.1 Hα Pn; Tool offset mode ON
:
:
G49; or H0; Tool offset cancel

Hα: Offset number is commanded with H-code.
If H-code is not commanded in G44.1 block, alarm PS0536, "ILLEGAL COMMAND IN G44.1" occurs.
Pn: The direction of imaginary tool nose is commanded.
If P is not commanded, the direction of imaginary tool nose is selected according to the angle of the tool nose rotation axis (A-axis) and the swivel head axis (B-axis) automatically.

Specify G44.1 without the data not related to tool offset.
If the posture of the tool is changed during G44.1 mode, the offset must be renewed by G44.1 command again.
The alarm PS0536, "ILLEGAL COMMAND IN G44.1" occurs in following cases.

(1) When G44.1 is not commanded after the posture of the tool has been changed, if the basic three axes (if the basic three axes are not set, the parallel axes) set to the parameter No. 1022 are moved.
(2) If the posture of the tool is changed at G44.1 block. (Sample program 1)
(3) If the posture of the tool is changed at the block in which basic three axes movement are commanded.
(4) When the machining center system is selected, if the offset number is specified by D code that is different from the offset number specified by H code in G44.1 mode. (Sample program 2)

Sample program 1 (Machining center system)

```
N10 G90 G00 X0.0 Y0.0 Z0.0;
N20 G44.1 H01;
N30 A180.0 B10.0;      ← The posture of the tool is changed.
N40 G44.1 H01;         ← G44.1 is commanded again.
N50 X10.0 Y10.0 Z10.0;
N60 A0.0 B20.0;        ← The posture of the tool is changed.
```

```
N70 X20.0 Y20.0 Z20.0;   ← Because G44.1 is not commanded,
                           PS0536 “ILLEGAL COMMAND IN G44.1” occurs.
N80 G49;
N90 M30;
```

Sample program 2 (Machining center system)

```
N10 G90 G00 X0.0 Y0.0 Z0.0;
N20 G44.1 H01;
N30 G17 G41 D01;   ← The offset number is the same as the offset number specified by H code.
N40 X10.0 Y10.0;
N50 G41.D03;       ← Because the offset number is different from the offset number
                     specified by H code, PS0536 “ILLEGAL COMMAND IN G44.1” occurs.
N60 X20.0 Y20.0;
N70 G40;
N80 G49;
N90 M30;
```

Explanation

Setting of basic machine composition

Before using this function, the basic machine composition must be set by related parameters.

The reference angle of the tool nose rotation axis (A-axis) in tool measuring position is selected by bit 0 (TRD) of the parameter No.19641 between 0 and 180 degree.

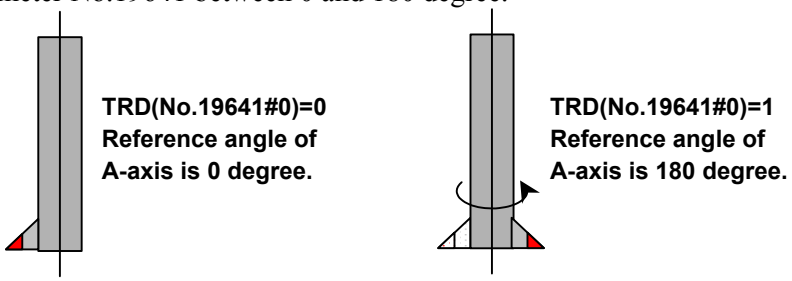


Fig.15.7 (b) Reference angle of the tool nose rotation axis (A-axis)

The reference angle of the swivel head axis (B-axis) in tool measuring position is set by the parameter No.19642.

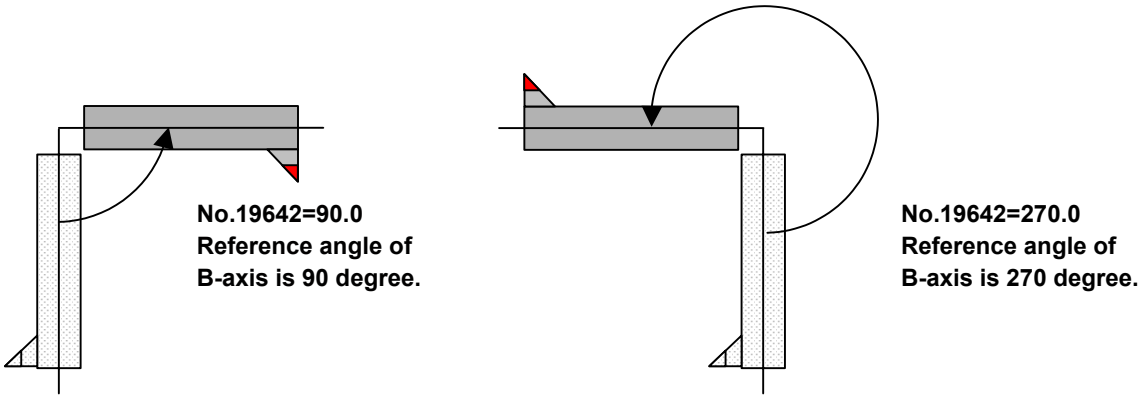


Fig.15.7 (c) Reference angle of the swivel head axis (B-axis)

The direction of rotation of swivel head axis (B-axis) is set by bit 1 (SRD) of the parameter No.19641.

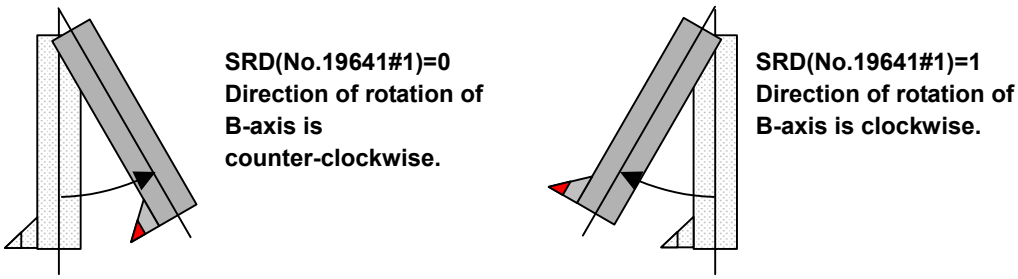


Fig.15.7 (d) Direction of rotation of swivel head axis (B-axis)

It is set by bit 2 (INW) of the parameter No.19641 whether the wear offset is taken into account in the offset calculation or not.
When the wear offset isn't taken into account in the offset calculation, an operator needs to measure the amount of wear after the tool is rotated and inputs it on the offset screen.

Change of the direction of imaginary tool nose by the rotation of each axis

The direction of imaginary tool nose is selected from 8 directions at intervals of 45-degree. The relation between imaginary tool nose number and the direction of imaginary tool nose is shown at Fig.15.7 (e).

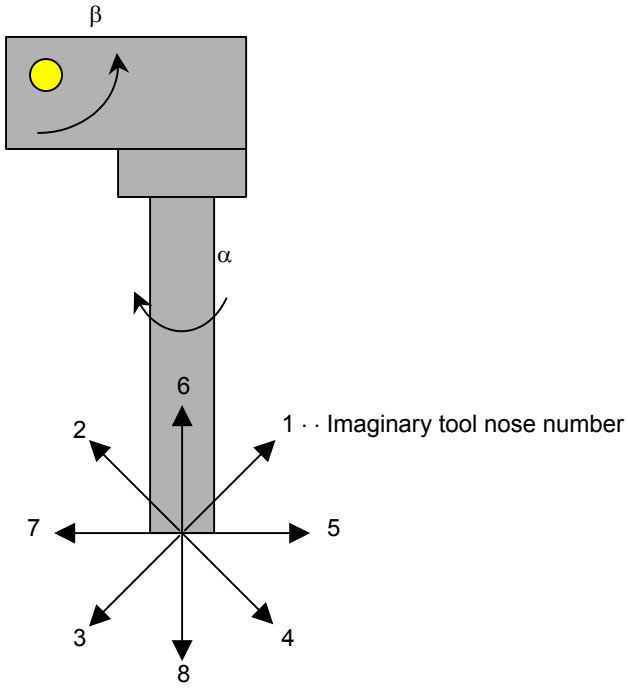


Fig.15.7 (e) Relationship between imaginary tool nose number and direction

Usually, in case that tool type is A in Fig.15.7 (f), the imaginary tool nose number is selected from 1 to 4. In case that tool type is B, the imaginary tool nose number is selected form 5 to 8.

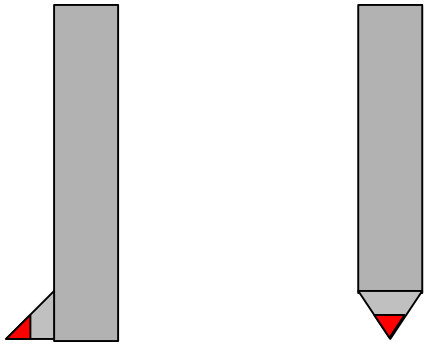


Fig.15.7 (f) Tool types

The swivel head axis (β) is approximated to 4 sections (90×n [degree] (n=1,2,3,4)). The approximation of angle of the swivel head axis (β) is shown at Fig.15.7 (g).

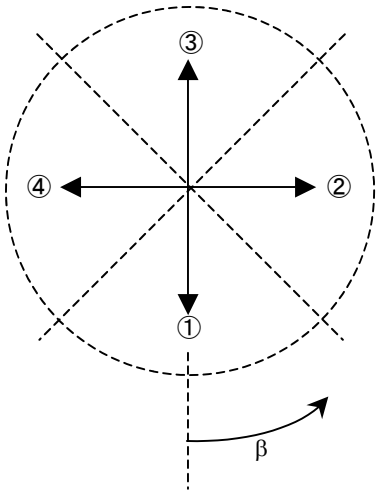


Fig.15.7 (g) The approximation of β

The commanded angle of the tool nose rotation axis α must be selected between 0 and 180 degrees.
When the commanded angle of the tool nose rotation axis α is 0.0≤α≤90.0, 270.0<α<360.0, α is approximated to 0.0.
When α is 90.0<α≤270.0, α is approximated to 180.0.
The relation among axis angle α, β and the direction of imaginary tool nose is shown at the Table15.7 (a) below.
When the imaginary tool nose direction is commanded by 1 to 4, the direction after α, β are rotated is selected from 1 to 4 automatically. When the direction is commanded by 5 to 8, the direction after α, β are rotated is selected from 5 to 8 automatically.

Table15.7 (a) Relationship among α, β and direction

β	α	Imaginary tool nose direction							
0.0≤(1)<45.0	0	1	2	3	4	5	6	7	8
	180	2	1	4	3	7	6	5	8
45.0≤(2)<135.0	0	2	3	4	1	6	7	8	5
	180	3	2	1	4	8	7	6	5
135.0≤(3)<225.0	0	3	4	1	2	7	8	5	6
	180	4	3	2	1	5	8	7	6
225.0≤(4)<315.0	0	4	1	2	3	8	5	6	7
	180	1	4	3	2	6	5	8	7

β	α	Imaginary tool nose direction							
315.0≤(1)<360.0	0	1	2	3	4	5	6	7	8
	180	2	1	4	3	7	6	5	8

When bit 1 (SRD) of the parameter No. 19641 is set to 1, if β=45.0 is specified, β angle is assumed to be 315.0 on calculation. Therefore, when bit 1 (SRD) of the parameter No. 19641 is set to 0, the imaginary tool nose direction is changed from (1) area to (2) area, however when bit 1 (SRD) of the parameter No. 19641 is set to 1, if β=45.0 is specified, the tool nose direction is not changed.

Example1 Change of tool nose compensation value (In case that tool type is A)

When the tool nose rotation axis and the swivel head axis are rotated as like Fig.15.7 (h), the direction of imaginary tool nose and the compensation value are changed as following. (The radius of tool nose is defined as R.)

	α=0,β=0	α=180,β=0	α=180,β= - 60
Imaginary tool nose number	3	4	3
Tool nose compensation value of X	Rx=+R	Rxnew=+R	Rxnew=+R
Tool nose compensation value of Z	Rz=+R	Rznew=-R	Rznew=+R

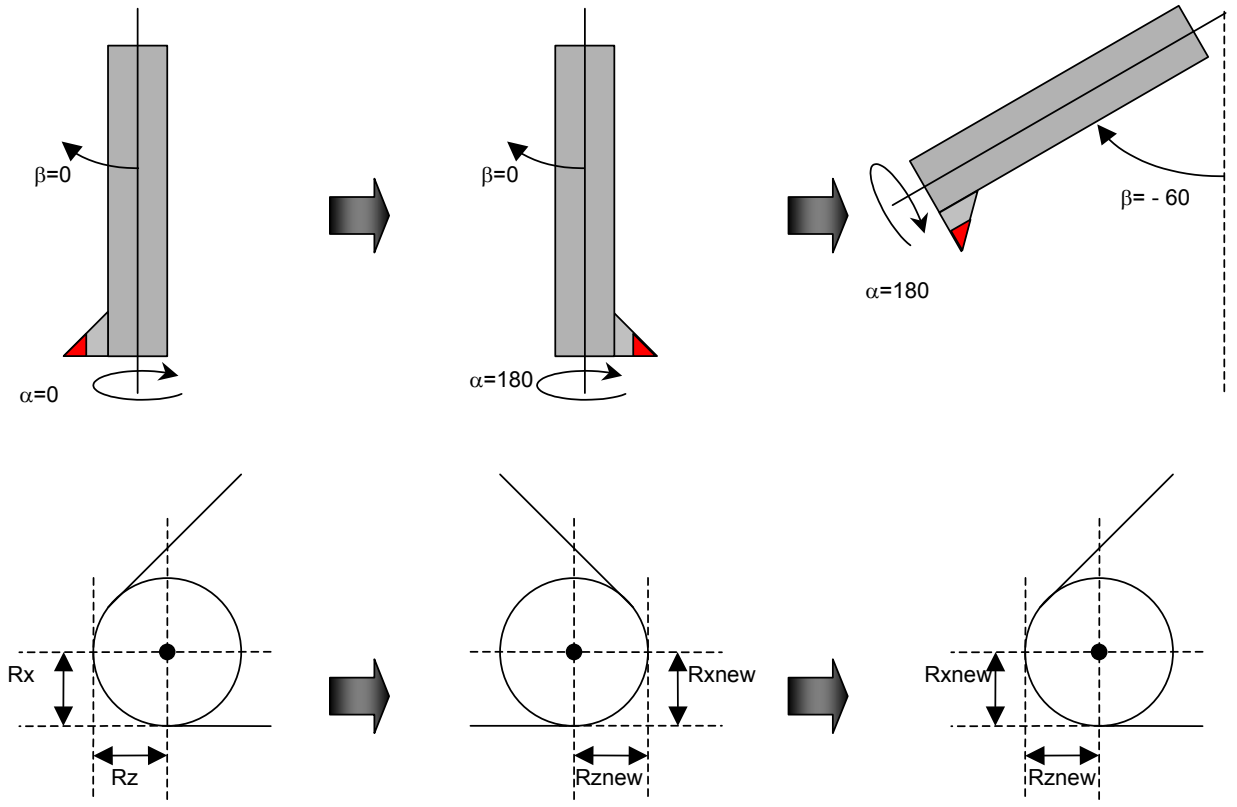


Fig.15.7 (h) Change of tool nose compensation value (A-type tool)

Example2 Change of tool nose compensation value (In case that tool type is B)

When the tool nose rotation axis and the swivel head axis are rotated as like
Fig.15.7 (i), the direction of imaginary tool nose and the compensation value are changed as following. (The radius of tool nose is defined as R.)

	α=0,β=0	α=180,β=0	α=180,β= - 60
Imaginary tool nose number	8	8	7
Tool nose compensation value of X	Rx=+R	Rxnew=+R	Rxnew=0
Tool nose compensation value of Z	Rz=0	Rznew=0	Rznew=+R

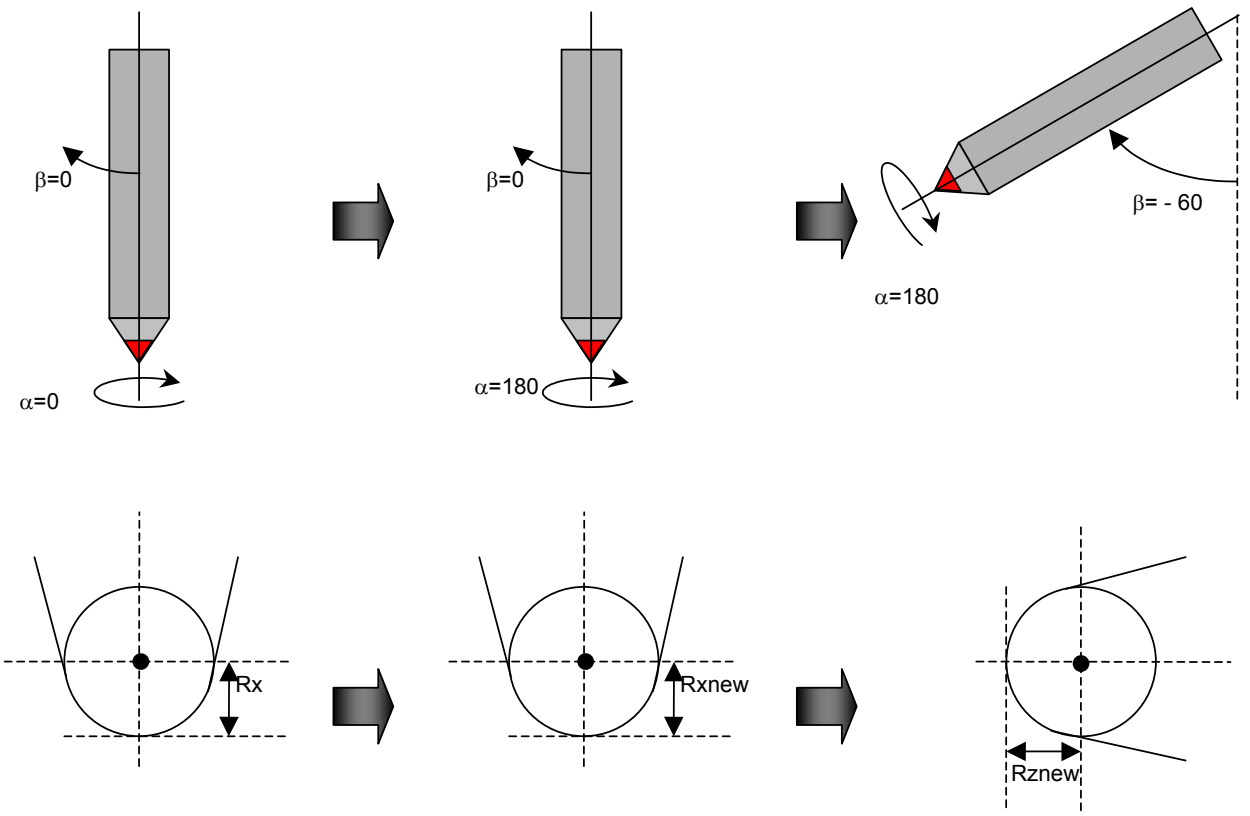


Fig.15.7 (i) Change of tool nose compensation value (B-type tool)

Example3 Change of the tool offset value

In this section, the calculation of tool offset of each axis (X, Z) is shown when the tool nose rotation axis and the swivel head axis are rotated as like Fig.15.7 (j).
At first, set the value below.
[Setting of the value]
Bit 0 (TRD) of the parameter No.19641 = 0
Bit 1 (SRD) of the parameter No.19641 = 0
Bit 2 (INW) of the parameter No.19641 = 0
Parameter No.19642 = 0.0
α=180.0degree, β= - 60.0degree
Pset_x=P19711+P19666=104.4452mm
Pset_z=P19709=-119.9922mm
offset_x=105.1030mm
offset_z=22.0580mm
Rx=Rz=2.0mm
The direction of imaginary tool nose is 3 when α=0, β=0. It can be known from Table15.7 (a) that the direction of imaginary tool nose after rotation of α and β is 3. Therefore Rxnew=2.0 and Rznew=2.0.

The tool offset of each axis after the rotation of α, β is calculated by each value above.
offset_x(After conversion)≒122.6159[mm]
offset_z(After conversion)≒231.7091[mm]

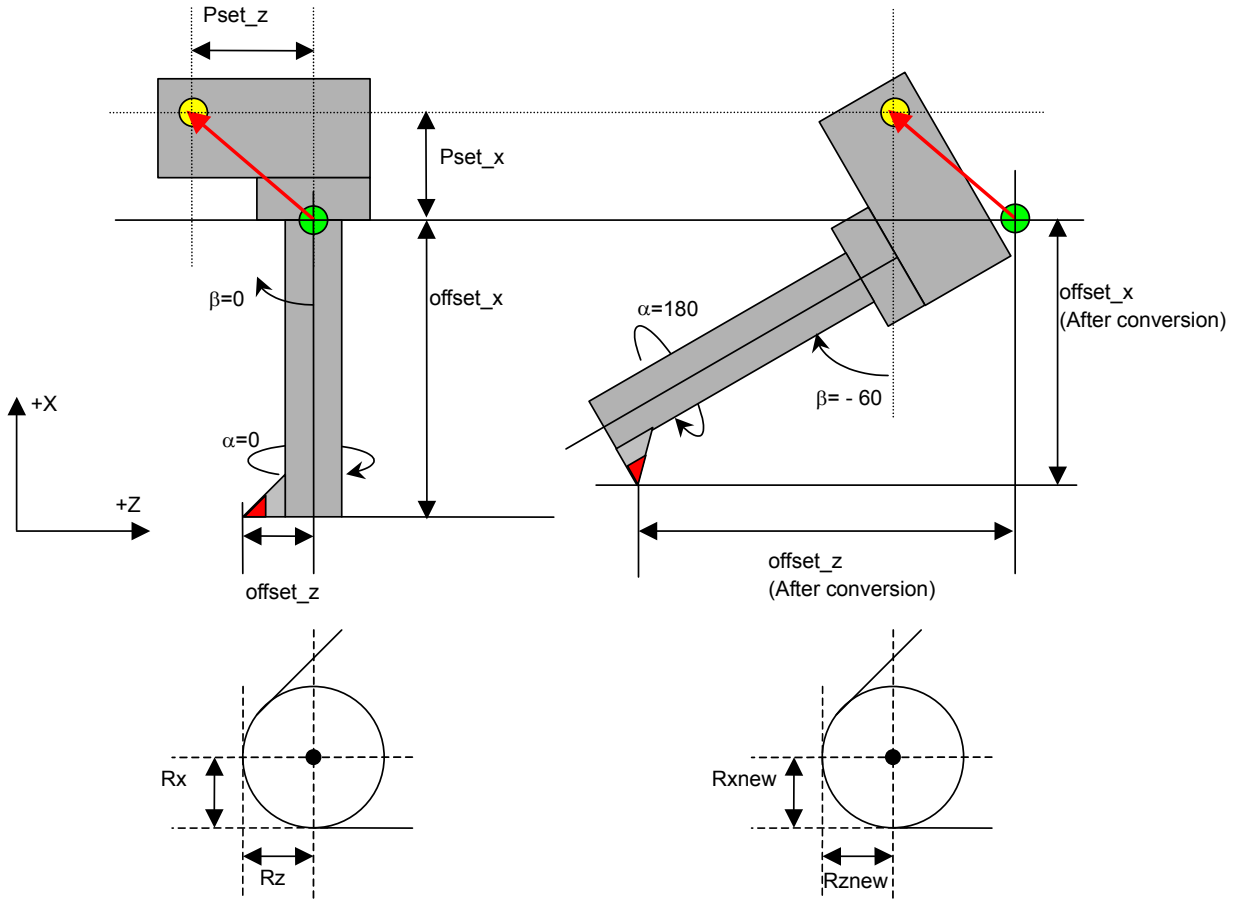


Fig.15.7 (j) Change of tool offset value by rotation of α and β

Setting of the rotation axis by parameter

The tool nose rotation axis and the swivel head axis are decided by the setting of the rotation axis selected by bits 0 (RS1) to 2 (RS3) of the parameter No.19640.

Setting of 1st rotation axis: the parameters No.19681, 19682
Setting of 2nd rotation axis: the parameters No.19686, 19687
Setting of 3rd rotation axis: the parameters No.19691, 19692

No.19640			Tool nose rotation axis	Swivel head axis
#2	#1	#0		
0	0	0	1st axis	2nd axis
0	0	1	1st axis	3rd axis
0	1	0	2nd axis	1st axis
0	1	1	2nd axis	3rd axis
1	0	0	3rd axis	1st axis
1	0	1	3rd axis	2nd axis
1	1	0		
1	1	1		

When (#2,#1,#0) = (1,1,0) or (1,1,1), the meaning of these parameters is the same as setting (0,0,0).(The tool nose rotation axis is the 1st rotation axis and the swivel head axis is the 2nd rotation axis.)

NOTE
The parameters No. 19691, 19692 that set the 3rd rotation axis are effective only in the Tool offset conversion function. In other 5-axis functions, these parameters are not effective.

The axis direction of the tool nose rotation axis must be set to X-axis or Z-axis and the axis direction of the swivel head axis must be set to Y-axis.
If the axis direction of the tool nose rotation axis is not set, the reference tool axis direction (Parameter No. 19697) is selected.
The axis that the converted offset is applied to X or Z of the basic three axes, or the axes parallel to X or Z specified by the parameter No. 1022. The offset value of Y-axis is always the same as the value set on the offset screen.

In the following cases, the alarm PS0535 "TL OFS CONVERSION SETTING ERROR" will occur when G44.1 is commanded.
- The swivel head axis is not set as rotation axis.
- The same rotation axis is set to tool nose rotation axis and swivel head axis.
- The axis direction of the tool nose rotation axis is not X-axis or Z-axis.
- The axis direction of the swivel head axis is not Y-axis.
- The number of the axis which converted offset is applied to is short.

The parameters No. 19682, 19687, 19691 that define the axis direction of the rotation axis have to be 1 or 3.
Setting value of the parameter No. 19697 that defines the reference tool axis direction is the same as the value of the tool nose rotation axis direction.
The parameters No. 19698, 19699 that define the angle when the reference tool axis direction is tilted are ignored in this function.

Information of the tool offset conversion

The information of tool offset calculated by G44.1 can be checked on diagnosis display. The meaning of diagnosis data is shown below.
- No.1800 : Reference angle of the tool nose rotation axis
- No.1801 : Reference angle of the swivel head axis
- No.1802 : Direction of imaginary tool nose before conversion
- No.1803 : Angle of the tool nose rotation axis for conversion
- No.1804 : Angle of the swivel head axis for conversion
- No.1805 : Direction of imaginary tool nose after conversion
- No.1806 : Offset value of X-axis before conversion
- No.1807 : Offset value of Y-axis before conversion
- No.1808 : Offset value of Z-axis before conversion
- No.1809 : Offset value of X-axis after conversion
- No.1810 : Offset value of Y-axis after conversion
- No.1811 : Offset value of Z-axis after conversion

Limitations

In this section, the matters that should be noted when this function is used are shown.

Illegal function in G44.1 mode

(1) If G44.1 is commanded in the mode shown below, the alarm PS0536, "ILLEGAL COMMAND IN G44.1" occurs. And if these functions are commanded in G44.1 mode, the alarm PS0536 "ILLEGAL COMMAND IN G44.1" occurs.
   - Tool center point control
   - Workpiece setting error compensation

(2) If the following G code are specified in G44.1 block, the alarm PS0536 "ILLEGAL COMMAND IN G44.1" occurs.
   - G20 (Input in inch)
   - G21 (Input in mm)

(3) When this function is used, a part of specification is limited to the following functions.
   - The protection of the tool offset in 8 level data protect function is invalid.

(4) The following functions cannot be used with this function.
- Tilted Working Plane Indexing

About measuring of the tool offset

The tool offset must be measured when the angle of the tool nose rotation axis and the swivel head axis are set to 0 degree.

The relation between tool length compensation and tool offset

T

When G44.1 command is specified in G43 mode, or G43 command is specified in G44.1 mode, the alarm PS0536 “ILLEGAL COMMAND IN G44.1” will occur.
When G44.1 command is specified in G43.7 mode, the tool offset that is specified in G43.7 mode is cancelled automatically, and the tool offset which is specified by G44.1 command is newly activated.
When G43.7 command is specified in G44.1 mode, the tool offset that is specified in G44.1 mode is cancelled automatically, and the tool offset which is specified by G43.7 command is newly activated.

M

When G44.1 command is specified in G43 mode or G43.7 mode, the tool offset which is specified in G43 mode or G43.7 mode is cancelled automatically, and the tool offset that is specified by G44.1 command is newly activated.
When G43 or G43.7 command is specified in G44.1 mode, the tool offset which is specified in G44.1 mode is cancelled automatically, and the tool offset which is specified by G43 or G43.7 command is newly activated.

Offset with tool movement and offset with coordinate shift

M

The value, which is the sum of geometry compensation and wear compensation, is applied to the tool compensation according to the type specified by the bit 6(TOS) of the parameter No.5006, bit 2(TOP) of the parameter No. 11400. It is not possible to compensate geometry offset and wear offset separately.

Parameters for tool offset

The parameters, which are effective and not effective in G44.1 mode, are shown below.

Effective parameter

Bit 6 (TOS) of the parameter No.5006
Bit 2 (TOP) of the parameter No.11400

T

Bit 2 (LWT) of the parameter No.5002
Bit 4 (LGT) of the parameter No.5002
Bit 5 (LGC) of the parameter No.5002
Bit 6 (LWM) of the parameter No.5002
Bit 7 (TGC) of the parameter No.5003
Bit 1 (ORC) of the parameter No.5004
Bit 3 (LVC) of the parameter No.5006
Bit 0 (OWD) of the parameter No.5040

M

Bit 6 (LVK) of the parameter No.5003
Bit 2 (ODI) of the parameter No.5004
Bit 6 (3OC) of the parameter No.5007
Bit 7 (3OF) of the parameter No.5007

Non-effective parameter

Bit 6 (EVO) of the parameter No.5001

The setting is fixed to "0: If a tool compensation value is modified in the offset mode (G43 or G44) The new value becomes valid in a block where G43, G44, or an H code is specified next." regardless of the setting value of this parameter.

T

Bit 1 (LGN) of the parameter No.5002

The setting is fixed to "0: Geometry offset number of tool offset is the same as wear offset number" regardless of the setting value of this parameter.

Bit 7 (WNP) of the parameter No.5002

This parameter is invalid because the geometry offset number and wear offset number cannot be specified independently in this function.

Bit 6 (3OC) of the parameter No.5007

The setting is fixed to "0: If tool length compensation is not cancelled before three-dimensional coordinate conversion is specified, an alarm is not raised." regardless of the setting value of this parameter.

Bit 7 (3OF) of the parameter No.5007

The setting is fixed to "0: If three-dimensional coordinate conversion is not nested with a command for tool length compensation, no alarm is issued." regardless of the setting value of this parameter.

M

Bits 0 (TLC) and 1 (TLB) of parameter No.5001

Tool offset is applied to X axis and Y axis regardless of the setting value of this parameter.

Bit 3 (TAL) of the parameter No.5001

The setting is fixed to "1: In tool length compensation C, any alarm dose not be generated even if two or more axes are offset" regardless of the setting value of this parameter.

Bit 4 (EVR) of the parameter No.5001

The setting is fixed to "0: When a tool compensation value is changed in tool radius - tool nose radius compensation mode, the new value becomes valid in a block where the next D or H code is specified.

Bit 5 (TIP) of parameter No.5009

The setting is fixed to "1:In cutter compensation or tool nose radius compensation, the virtual tool tip direction is used." regardless of the setting value of this parameter.

About combination with the flexible path axis assignment function

When the rotation axis number set as tool nose rotation axis or swivel head axis is changed, set the value of parameters No. 19681, 19686, 19691 by G10 command.

The rotation axis set as tool nose rotation axis or swivel head axis must be in the same path.

Diagnosis display

Diagnosis	1800	Reference angle of the tool nose rotation axis

[Data type] Real path

[Unit of data] degree (machine unit)

[Valid data range] 0.0 to 360.0

The reference angle of the tool nose rotation axis for the tool offset conversion is displayed.

This angle depends on the setting of bit 0 (TRD) of the parameter No. 19641.

Diagnosis	1801	Reference angle of the swivel head axis

[Data type] Real path
[Unit of data] degree (machine unit)
[Valid data range] 0.0 to 360.0
The reference angle of the swivel head axis for the tool offset conversion is displayed.
This angle depends on the setting of the parameter No. 19642.
The direction of rotation depends on the setting of bit 1 (SRD) of the parameter No. 19641.

Diagnosis	1802	Direction of imaginary tool nose before conversion

[Data type] Byte path
[Unit of data] None
[Valid data range] 0 to 8
The direction of imaginary tool nose before the tool offset conversion is displayed.
This imaginary tool nose number is set to offset screen for each tool.

Diagnosis	1803	Angle of the tool nose rotation axis for conversion

[Data type] Real path
[Unit of data] degree (machine unit)
[Valid data range] 0.0 to 360.0
The angle of the tool nose rotation axis for the tool offset conversion is displayed.
This angle depends on the commanded angle and the setting of bit 0 (TRD) of the parameter No. 19641.

Diagnosis	1804	Angle of the swivel head axis for conversion

[Data type] Real path
[Unit of data] degree (machine unit)
[Valid data range] 0.0 to 360.0
The angle of the swivel head axis for the tool offset conversion is displayed.
This angle depends on the commanded angle and the setting of the parameter No. 19642.
The direction of rotation depends on the setting of bit 1 (SRD) of the parameter No. 19641.

Diagnosis	1805	Direction of imaginary tool nose after conversion

[Data type] Byte path
[Unit of data] None
[Valid data range] 0 to 8
The direction of imaginary tool nose after the tool offset conversion is displayed.
If the direction of imaginary tool nose did not be commanded in G44.1 block, the direction is converted by angle of the tool nose rotation axis and the swivel head axis automatically.

Diagnosis	1806	Offset value of X-axis before conversion

[Data type] Real path
[Unit of data] mm, inch (offset unit)
The offset value of X-axis before the tool offset conversion is displayed.
This value is the same as the sum of geometry and wear offset value for “X” set on offset screen.

Diagnosis	1807	Offset value of Y-axis before conversion

[Data type] Real path
[Unit of data] mm, inch (offset unit)
The offset value of Y-axis before the tool offset conversion is displayed.
This value is the same as the sum of geometry and wear offset value for “Y” set on offset screen.

Diagnosis	1808	Offset value of Z-axis before conversion

[Data type] Real path
[Unit of data] mm, inch (offset unit)
The offset value of Z-axis before the tool offset conversion is displayed.
This value is the same as the sum of geometry and wear offset value for “Z/LENGTH” set on offset screen.

Diagnosis	1809	Offset value of X-axis after conversion

[Data type] Real path
[Unit of data] mm, inch (offset unit)
The offset value of X-axis after the tool offset conversion by G44.1 is displayed.
When bit 2 (INW) of the parameter No. 19641 is set to 0, the sum of geometry conversion value and wear conversion value is displayed.
When bit 2 (INW) of the parameter No. 19641 is set to 1, the sum of geometry conversion value and wear setting value is displayed.
This value takes the place of geometry and wear offset value for “X” set on offset screen.

Diagnosis	1810	Offset value of Y-axis after conversion

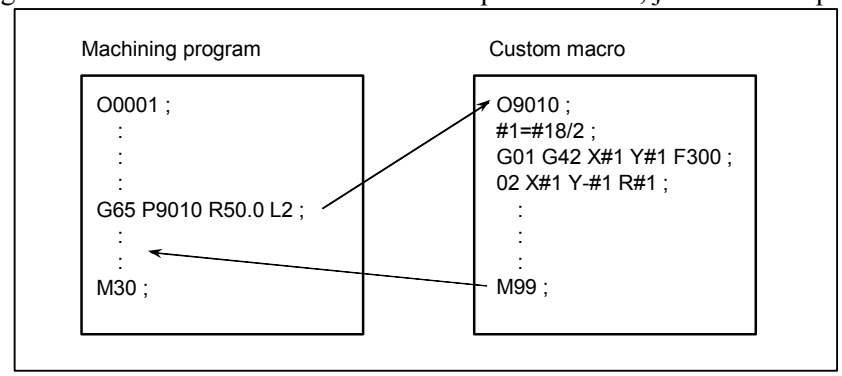
[Data type] Real path
[Unit of data] mm, inch (offset unit)
The offset value of Y-axis after the tool offset conversion by G44.1 is displayed.
When bit 2 (INW) of the parameter No. 19641 is set to 0, the sum of geometry conversion value and wear conversion value is displayed.
When bit 2 (INW) of the parameter No. 19641 is set to 1, the sum of geometry conversion value and wear setting value is displayed.
This value takes the place of geometry and wear offset value for “Y” set on offset screen.

Diagnosis	1811	Offset value of Z-axis after conversion

[Data type] Real path
[Unit of data] mm, inch (offset unit)
The offset value of Z-axis after the tool offset conversion by G44.1 is displayed.
When bit 2 (INW) of the parameter No. 19641 is set to 0, the sum of geometry conversion value and wear conversion value is displayed.
When bit 2 (INW) of the parameter No. 19641 is set to 1, the sum of geometry conversion value and wear setting value is displayed.
This value takes the place of geometry and wear offset value for “Z/LENGTH” set on offset screen.

16 CUSTOM MACRO

Although subprograms are useful for repeating the same operation, the custom macro function also allows use of variables, arithmetic and logic operations, and conditional branches for easy development of general programs such as pocketing and user-defined canned cycles.
A machining program can call a custom macro with a simple command, just like a subprogram.



16.1 VARIABLES

An ordinary machining program specifies a G code and the travel distance directly with a numeric value; examples are G100 X100.0.
With a custom macro, numeric values can be specified directly or using a variable number. When a variable number is used, the variable value can be changed by a program or using operations on the MDI panel.

```
#1=#2+100 ;
G01 X#1 F300 ;
```

Explanation

- Variable representation

When specifying a variable, specify a number sign (#) followed by a variable number.
#i (i = 1, 2, 3, 4, ...)
[Example] #5
#109
#1005
A variable can also be represented as follows using <expression> described in the section about arithmetic and logic operation commands.
#[<expression>]
[Example] #[#100]
#[#1001-1]
#[#6/2]
Variable #i shown in the following can be replaced with a variable of #[<expression>].

- Types of variables

Variables can be classified as local variables, common variables, and system variables according to the variable number. Each of those variables has its own usage and characteristics. Read-only system constants are also provided.

- Range of variable values

Local and common variables can have a value in the following ranges. If the result of calculation exceeds the range, an alarm PS0111, "OVERFLOW :FLOATING" is issued.

When bit 0 (F16) of parameter No.6008 = 0

Maximum value: approx. $\pm 10^{308}$

Minimum value: approx. $\pm 10^{-308}$

Numeric data handled by a custom macro conforms to the IEEE standard and is handled as a double-precision real number. An error resulting from operation depends on the precision.

When bit 0 (F16) of parameter No.6008 = 1

Maximum value: approx. $\pm 10^{47}$

Minimum value: approx. $\pm 10^{-29}$

- Local variable (#1-#33)

A local variable is a variable that is used locally in a macro. That is, local variable #i used by a macro called at a certain time is different from that used by a macro called at another time, regardless of whether the two macros are the same. Therefore, for example, when macro A calls macro B during multiple calls or the like, it is impossible for macro B to corrupt a local variable used by macro A by erroneously using the variable.

A local variable is used to pass arguments. For information on correspondence between arguments and addresses, see the section about macro calling commands. The initial state of a local variable to which no arguments are passed is <null> and the user can freely use the variable. The attribute of a local variable is READ/WRITE enabled.

- Common variable (#100-#199, #500-#999)

A common variable is shared among the main program, subprograms called by the main program, and macros while a local variable is used locally in a macro. That is, #i used by a macro is the same as that used by another macro. Therefore, a resultant common variable obtained by using a macro can be used by another macro. The attribute of a common variable is basically READ/WRITE enabled. However, the common variable can be protected (its attribute is set to READ only) by specifying its variable number using parameters Nos.6031 and 6032. A common variable can be freely used by the user even when its usage is not defined by the system. The number of common variables can be specified by selecting one of the following options.

(a) 100 common variables (specified only with the custom macro option)

The common variables #100 to #149 and #500 to #549 can be used. The variables #100 to #149 are cleared during power-down, but the variables #500 to #549 are retained during power-down.

(b) 600 common variables (specified with the custom macro option or custom macro common variable addition option)

The common variables #100 to #199 and #500 to #999 can be used. The variables #100 to #199 are cleared during power-down, but the variables #500 to #999 are retained during power-down.

- Write protection of a common variable

Multiple common variables (#500 to #999) can be protected (their attributes are set to READ only) by setting variable numbers in parameters Nos.6031 and 6032. This protection is enabled for both Input/All Clear by MDI on the macro screen and write operation by a macro program. If the NC program specifies WRITE operation (used in the left side) for a common program in the set range, an alarm PS0116, "WRITE PROTECTED VARIABLE" is issued.

- Custom macro common variables between each path

Some or all of the custom macro common variables (#100 to #149 (#199, #499) and #500 to #599 (#999)) can be made common to all paths by setting parameter No. 6036 (#100 to #149 (#199, #499)) and parameter No. 6037 (#500 to #599 (#999)) appropriately. Also, whether or not to make the variables common can be selected on a path-by-path basis by setting bit 0 (NC1) of parameter No. 6020 for parameter No. 6036 and bit 1 (NC2) of parameter No. 6020 for parameter No. 6037 appropriately.

[Example]

Assume that system common parameter No. 6036 is set to 20.

If the setting of parameter No. 6036 is not to be reflected in the fourth path only, make the settings below (Table 16.1 (a)).

Table 16.1 (a)

Path number	No.6036	NC1	Area of the custom macro variables used
1	20	0	Variables #100 to #119 are common to all paths, while other variables for each path are independent.
2		0	
3		0	
4		1	Variables for path 4 are all independent.

- System variable

A variable whose usage does not vary in the system. The attribute of a system variable is READ only, WRITE only, or READ/WRITE enabled depending on the nature of a system variable.

- System constant

A system constant can be referenced as with a variable even though its value is fixed. The attribute of a system constant is READ only.

- Omission of the decimal point

When a variable value is defined in a program, the decimal point can be omitted.

[Example] When #1 = 123; is defined, the actual value of variable #1 is 123.000.

- Referencing variables

The value following an address can be replaced with a variable. When programming as <address>#i or <address>-#i, the variable value or the complement of it is used as the specified value of the address.

[Example] F#33 is the same as F1.5 when #33 = 1.5.
Z-#18 is the same as Z-20.0 when #18 = 20.0.
G#130 is the same as G3 when #130 = 3.0.
A variable cannot be referenced using address/, :, or O and N.

[Example] Programming such as O#27 or N#1 is not allowed.
n (n = 1 to 9) in the optional block skip /n cannot be a variable.
A variable number cannot be specified by a direct variable.

[Example] When replacing 5 in #5 with #30, specify #[#30] instead of ##30. No values exceeding the maximum allowable value for each address can be specified.

[Example] When #140 = 120, G#140 exceeds the maximum allowable value.
When a variable is used as address data, the variable is automatically rounded off to the number of significant figures of each address or less.

[Example] For a machine with an increment system of 1/1000 mm (IS-B), when #1 = 12.3456, G00 X#1; becomes G00 X12.346;.
If <expression>, described later, is used, the value following an address can be replaced with <expression>.
<address>[<expression>] or <address>-[<expression>]
The program code shown above indicates the value of <expression> or the complement of the value is used as an address value. Note that a constant with no decimal point, enclosed in brackets ([ ]), is assumed to have a decimal point at the end.

[Example] X[#24+#18*COS[#1]]
Z-[#18+#26]

- Undefined variable

When the value of a variable is not defined, such a variable is referred to as a "null" variable. Variables #0 and #3100 are always null variables. They cannot be written to, but they can be read.

(a) Quotation
When an undefined variable is quoted, the address itself is also ignored.

Original command	G90 X100 Y#1
Equivalent command when #1 = <null>	G90 X100
Equivalent command when #1 = 0	G90 X100 Y0

(b) Definition/replacement, addition, multiplication
When a local variable or common variable is directly replaced with <null>, the result is <null>. When a system variable is directly replaced with <null> or the result of calculation including <null> is replaced, a variable value of 0 is assumed.

Original expression (local variable)	#2=#1	#2=#1*5	#2=#1+#1
Replacement result (when #1 = <null>)	<null>	0	0
Replacement result (when #1 = 0)	0	0	0

Original expression (common variable)	#100=#1	#100=#1*5	#100=#1+#1
Replacement result (when #1 = <null>)	<null>	0	0
Replacement result (when #1 = 0)	0	0	0

Original expression (system variable)	#2001=#1	#2001=#1*5	#2001=#1+#1
Replacement result (when #1 = <null>)	0	0	0
Replacement result (when #1 = 0)	0	0	0

(c) Comparison
<null> differs from 0 only for EQ and NE.
<null> is equal to 0 for GE, GT, LE, and LT.

- When <null> is assigned to #1

Conditional expression	#1 EQ #0	#1 NE 0	#1 GE #0	#1 GT 0	#1 LE #0	#1 LT 0
Evaluation result	Established (true)	Established (true)	Established (true)	Not established (false)	Established (true)	Not established (false)

- When 0 is assigned to #1

Conditional expression	#1 EQ #0	#1 NE 0	#1 GE #0	#1 GT 0	#1 LE #0	#1 LT 0
Evaluation result	Not established (false)	Not established (false)	Established (true)	Not established (false)	Established (true)	Not established (false)

- Specifying a system variable (constant) by its name

A system variable (constant) is specified by its variable number, but it can also be specified by its predetermined system variable (constant) name. A system variable (constant) name begins with an underscore (_), followed by up to seven uppercase letters, numerics, or underscores. For axis-dependent variables (such as coordinates) or variables having a lot of data of similar types (such as tool compensation), subscript [n] (n: integer) can be used to specify values. In this case, n can be specified in <expression> format (calculation format).
The command format must be specified in [#system-variable-name] format, as shown below.

```
  [#_DATE]
[Example]
    [#_DATE]=20040117 ;        : 2004.01.17 is assigned to #3011 (year month date).
    [#_TIME]=161705 ;          : 16:17:05 is assigned to #3012 (time minute second).
    #101=[#_ABSMT[1]] ;        : #5021 (machine coordinate value of the 1st axis) is read off
                                 and assigned to #101.
```

#102=[#_ABSKP[#500*2]] ; : #506x (skip position of [#500*2]th axis) is read off and assigned to #102.

If a value other than an integer is specified for subscript n, a variable value is referenced, assuming that the fractional portion is rounded off.

[Example]

[#_ABSIO[1.4999999]] : This value is assumed to be [#_ABSIO[1]], that is, #5001.
[#_ABSIO[1.5000000]] : This value is assumed to be [#_ABSIO[2]], that is, #5002.

NOTE
1 When the specified variable name is not registered, an alarm PS1098, "NO VARIABLE NAME" is issued.
2 When a negative or other invalid subscript is specified, an alarm PS1099, "ILLLEGAL SUFFIX [ ]" is issued.

- System constant #0, #3100-#3102 (Attribute: R)

Constants used as fixed values in the system can be used as system variables. Such constants are called system constants. The system constants provided are shown below.

Constant number	Constant name	Description
#0, #3100	[#_EMPTY]	Null
#3101	[#_PI]	Circular constant π = 3.14159265358979323846
#3102	[#_E]	Base of natural logarithm e = 2.71828182845904523536

- Specifying a common variable by its name

Specifying a variable name set by the SETVN command described later allows reading from or writing to a common variable.

The command must be specified in the form [#common-variable-name] such as [#VAR500].

[Example]

X[#POS1] Y[#POS2] ; : Specifying a position by the variable name
[#POS1] = #100+#101 ; : Executing a assignment statement by the variable name
#[100+[#ABS]] = 500 ; : Same as above (by a variable number)
#500 = [1000+[#POS2]*10] ; : Reading a variable by a variable name

- Setting and specifying the name of a common variable (SETVN)

For the 50 common variables, #500 to #549, a name of up to eight characters can be specified by using a command as shown below. If the custom macro variable name expansion function is enabled, a name of up to 31 characters can be specified.

SETVN n [VAR500, VAR501, VAR502,.....] ;

n represents the starting number of a common variable for which the name is specified.

VAR500 is the variable name of variable n, VAR501 is the variable name of variable n+1, and VAR502 is the variable name of variable number n+2, and so on. Each string is delimited by a comma (,). All codes that can be used as meaningful information in a program except control in, control out, [,], EOB, EOR, and : (colon in a program number) can be used. However, each name must begin with an alphabetical character. Variable names are not cleared on switch-off.

Specifying a set variable name allows reading from or writing to the common variable. The command must be specified in the form [#common-variable-name] such as [#VAR500].

[Example] SETVN 510[TOOL_NO, WORK_NO, COUNTER1, COUNTER2];
The command above names the variables as follows.

Variable	Name
#510	TOOL_NO
#511	WORK_NO
#512	COUNTER1
#513	COUNTER2

The names specified by the command can be used in a program. For example, when 10 is assigned to #510, the expression [#TOOL_NO]=10; can be used instead #510=10;.
If the custom macro variable name expansion function is enabled,
the SETVN 510[TOOL_NUMBER000001_CHANGE_DETECT]; command assigns a name of 31 characters to the variable.

Variable	Name
#510	TOOL_NUMBER000001_CHANGE_DETECT

NOTE
If the same name was specified for different common variables, only the variable which has the smaller variable number can be referenced with the specified name.

16.2 SYSTEM VARIABLES

System variables can be used to read and write internal CNC data such as tool compensation values and current position data. System variables are essential for automation and general-purpose program development.

List of system variables and constants

n represents a subscript.
R, W, and R/W are attributes of a variable and indicate read-only, write-only, and read/write enabled, respectively.

- Interface signals

System variable number	System variable name	Attribute	Description
#1000-#1031	[#_UI[n]]	R	Interface input signals (BIT), UI000-UI031 NOTE) Subscript n represents a BIT position (0-31).
#1032-#1035	[#_UIL[n]]	R	Interface input signals (LONG), UI000-UI031/ UI100-UI131/ UI200-UI231/UI300-UI331 NOTE) Subscript n (0-3): 0 = UI000-UI031, 1 = UI100-UI131, 2 = UI200-231, 3 = UI300-UI331
#1036-#1067	[#_UI[n]]	R	Interface input signals (BIT), UI400-UI431 NOTE) Subscript n represents a BIT position (32-63).
#1068-#1071	[#_UIL[n]]	R	Interface input signals (LONG), UI400-UI431 / UI500-UI531 / UI600-UI631 / UI700-UI731 NOTE) Subscript n (4-7): 4=UI400-UI431, 5=UI500-UI531, 6=UI600-UI631, 7=UI700-UI731
#1100-#1131	[#_UO[n]]	R/W	Interface output signals (BIT), UO000-UO031 NOTE) Subscript n represents a BIT position (0-31).
#1132-#1135	[#_UOL[n]]	R/W	Interface output signals (LONG), UO000-UO031/ UO100-UO131/UO200-UO231/UO300-UO331 NOTE) Subscript n (0-3): 0 = UO000-UO031, 1 = UO100-UO131, 2 = UO200-231, 3 = UO300-UO331
#1136-#1167	[#_UO[n]]	R/W	Interface output signals (BIT), UO400-UO431 NOTE) Subscript n represents a BIT position (32-63).
#1168-#1171	[#_UOL[4]]-[#_UOL[7]]	R/W	Interface output signals (LONG), UO400-UO431/ UO500-UO531 / UO600-UO631 / UO700-UO731 NOTE) Subscript n (4-7): 4=UO400-UO431, 5=UO500-UO531, 6=UO600-UO631, 7=UO700-UO731

- Tool compensation value

M

Tool compensation memory A

System variable number	System variable name	Attribute	Description
#2001-#2200 #10001-#10999	[#_OFS[n]]	R/W	Tool compensation value Note) Subscript n represents a compensation number (1 to 200). When the number of sets is larger than 200, the numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).

Tool compensation memory B when bit 3 (V15) of parameter No.6000 = 0

System variable number	System variable name	Attribute	Description
#2001-#2200	[#_OFSW[n]]	R/W	Tool compensation value (wear) Note) Subscript n represents a compensation number (1 to 200).
#10001-#10999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2400	[#_OFSG[n]]	R/W	Tool compensation value (H code, geometry) Note) Subscript n represents a compensation number (1 to 200).
#11001-#11999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).

Tool compensation memory B when bit 3 (V15) of parameter No.6000 = 1

System variable number	System variable name	Attribute	Description
#2001-#2200	[#_OFSG[n]]	R/W	Tool compensation value (geometry) Note) Subscript n represents a compensation number (1 to 200).
#10001-#10999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2400	[#_OFSW[n]]	R/W	Tool compensation value (H code, wear) Note) Subscript n represents a compensation number (1 to 200).
#11001-#11999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).

Tool compensation memory C when bit 3 (V15) of parameter No.6000 = 0

System variable number	System variable name	Attribute	Description
#2001-#2200	[#_OFSHW[n]]	R/W	Tool compensation value (H code, wear) Note) Subscript n represents a compensation number (1 to 200).
#10001-#10999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2400	[#_OFSHG[n]]	R/W	Tool compensation value (H code, geometry) Note) Subscript n represents a compensation number (1 to 200).
#11001-#11999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#12001-#12999	[#_OFSDW[n]]	R/W	Tool compensation value (D code, wear) Note) Subscript n represents a compensation number (1 to 999).
# 13001-#13999	[#_OFSDG[n]]	R/W	Tool compensation value (D code, geometry) Note) Subscript n represents a compensation number (1 to 999).

Tool compensation memory C when bit 3 (V15) of parameter No.6000 = 1

System variable number	System variable name	Attribute	Description
#2001-#2200	[#_OFSHG[n]]	R/W	Tool compensation value (H code, geometry) Note) Subscript n represents a compensation number (1 to 200).
#10001-#10999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2400	[#_OFSHW[n]]	R/W	Tool compensation value (H code, wear) Note) Subscript n represents a compensation number (1 to 200).
#11001-#11999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2401-#2600	[#_OFSDG[n]]	R/W	Tool compensation value (D code, geometry) (Note 1) Subscript n represents a compensation number (1 to 200). Note 1) Enabled when bit 5 (D15) of parameter No.6004 = 1.
#12001-#12999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2601-#2800	[#_OFSDW[n]]	R/W	Tool compensation value (D code, wear) (Note 1) Subscript n represents a compensation number (1 to 200). Note 1) Enabled when bit 5 (D15) of parameter No.6004 = 1.
#13001-#13999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).

Tool compensation memory C
System variables not dependent on bit 3 (V15) of parameter No. 6000

System variable number	System variable name	Attribute	Description
#21001-#21999	[#_CORR_G[n]]	R/W	Corner R offset (geometry) Note) Subscript n represents a compensation number (1 to 999).
#22001-#22999	[#_CORR_W[n]]	R/W	Corner R offset (wear) Note) Subscript n represents a compensation number (1 to 999).

- Tool compensation value

T

Without tool geometry/wear compensation memory

System variable number	System variable name	Attribute	Description
#2001-#2064 #10001-#10999	[#_OFSX[n]]	R/W	X-axis compensation value (*1) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2101-#2164 #11001-#11999	[#_OFSZ[n]]	R/W	Z-axis compensation value (*1) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2264 #12001-#12999	[#_OFSR[n]]	R/W	Tool nose radius compensation value Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2301-#2364 #13001-#13999	[#_OFST[n]]	R/W	Virtual tool tip T position Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).

System variable number	System variable name	Attribute	Description
#2401-#2449 #14001-#14999	[#_OFSY[n]]	R/W	Y-axis compensation value (*1) Note) Subscript n represents a compensation number (1 to 49). When the number of sets is larger than 49, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

With tool geometry/wear compensation memory

System variable number	System variable name	Attribute	Description
#2001-#2064 #10001-#10999	[#_OFSXW[n]]	R/W	X-axis compensation value (wear) (*1) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2101-#2164 #11001-#11999	[#_OFSZW[n]]	R/W	Z-axis compensation value (wear) (*1) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2264 #12001-#12999	[#_OFSRW[n]]	R/W	Tool nose radius compensation value (wear) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2301-#2364 #13001-#13999	[#_OFST[n]]	R/W	Virtual tool tip T position Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2401-#2449 #14001-#14999	[#_OFSYW[n]]	R/W	Y-axis compensation value (wear) (*1) Note) Subscript n represents a compensation number (1 to 49). When the number of sets is larger than 49, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2451-#2499 #19001-#19999	[#_OFSYG[n]]	R/W	Y-axis compensation value (geometry) (*1) Note) Subscript n represents a compensation number (1 to 49). When the number of sets is larger than 49, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2701-#2749 #15001-#15999	[#_OFSXG[n]]	R/W	X-axis compensation value (geometry) (*1) Note) Subscript n represents a compensation number (1 to 49). When the number of sets is larger than 49, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 49).
#2801-#2849 #16001-#16999	[#_OFSZG[n]]	R/W	Z-axis compensation value (geometry) (*1) Note) Subscript n represents a compensation number (1 to 49). When the number of sets is larger than 49, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).
#2901-#2964 #17001-#17999	[#_OFSRG[n]]	R/W	Tool nose radius compensation value (geometry) Note) Subscript n represents a compensation number (1 to 64). When the number of sets is larger than 64, the numbers to the left can be used. Note) Subscript n represents a compensation number (1 to 999).

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- Tool compensation value when the complex machining tool offset function is enabled

M

When bit 3 (V15) of parameter No.6000 = 0

System variable number	System variable name	Attribute	Description
#2001-#2200	[#_OFSHW[n]] or [#_OFSZW[n]]	R/W	Z-axis tool offset (wear) (Old name: Tool compensation value (H code, wear)) Note) Subscript n represents a compensation number (1 to 200).
#10001-#10999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2400	[#_OFSHG[n]] or [#_OFSZG[n]]	R/W	Z-axis tool offset (geometry) (Old name: Tool compensation value (H code, geometry) Note) Subscript n represents a compensation number (1 to 200).
#11001-#11999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#12001-#12999	[#_OFSDW[n]] or [#_OFSRW[n]]	R/W	Tool radius compensation (wear) (Old name: Tool compensation value (D code, wear)) Note) Subscript n represents a compensation number (1 to 999).
#13001-#13999	[#_OFSDG[n]] or [#_OFSRG[n]]	R/W	Tool radius compensation (geometry) (Old name: Tool compensation (D code, geometry)) Note) Subscript n represents a compensation number (1 to 999).

When bit 3 (V15) of parameter No.6000 = 1

System variable number	System variable name	Attribute	Description
#2001-#2200	[#_OFSHG[n]] or [#_OFSZG[n]]	R/W	Z-axis tool offset (geometry) (Old name: Tool compensation value (H code, geometry)) Note) Subscript n represents a compensation number (1 to 200).
#10001-#10999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2201-#2400	[#_OFSHW[n]] or [#_OFSZW[n]]	R/W	Z-axis tool offset (wear) (Old name: Tool compensation value (H code, wear)) Note) Subscript n represents a compensation number (1 to 200).
#11001-#11999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2401-#2600	[#_OFSDG[n]] or [#_OFSRG[n]]	R/W	Tool radius compensation (geometry) (Note 1) (Old name: Tool compensation value (D code, geometry)) Subscript n represents a compensation number (1 to 200). Note 1: Enabled when bit 5 (D15) of parameter No.6004 = 1.
#12001-#12999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).
#2601-#2800	[#_OFSDW[n]] or [#_OFSRW[n]]	R/W	Tool radius compensation (wear) (Note 1) (Old name: Tool compensation value (D code, wear)) Subscript n represents a compensation number (1 to 200). Note 1: Enabled when bit 5 (D15) of parameter No.6004 = 1.
#13001-#13999			The numbers to the left can also be used. Note) Subscript n represents a compensation number (1 to 999).

System variables not dependent on bit 3 (V15) of parameter No. 6000

System variable number	System variable name	Attribute	Description
#21001-#21999	[#_CORR_G[n]]	R/W	Corner R offset (geometry) Note) Subscript n represents a compensation number (1 to 999).
#22001-#22999	[#_CORR_W[n]]	R/W	Corner R offset (wear) Note) Subscript n represents a compensation number (1 to 999).
#23001-#23999	[#_OFST[n]]	R/W	Virtual tool tip T position Note) Subscript n represents a compensation number (1 to 999).
#135001-#135999	[#_OFSXW[n]]	R/W	X axis tool offset (wear) Note) Subscript n represents a compensation number (1 to 999).
#136001-#136999	[#_OFSYW[n]]	R/W	Y axis tool offset (wear) Note) Subscript n represents a compensation number (1 to 999).
#137001-#137999	[#_OFSXG[n]]	R/W	X axis tool offset (geometry)) Note) Subscript n represents a compensation number (1 to 999).
#138001-#138999	[#_OFSYG[n]]	R/W	Y axis tool offset (geometry) Note) Subscript n represents a compensation number (1 to 999).

T

The same rules as those for conventional custom macro system variables apply.

- Workpiece coordinate system shift amount

T

System variable number	System variable name	Attribute	Description
#2501	[#_WKSFTX]	R/W	X-axis workpiece shift amount
#2601	[#_WKSFTZ]	R/W	Z-axis workpiece shift amount
#100751-#100800	[#_WZ_SFT [n]]	R/W	nth axis workpiece shift amount Note) Subscript n represents an axis number (1 to 50).

X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes

- Automatic operation or the like

System variable number	System variable name	Attribute	Description
#3000	[#_ALM]	W	Macro alarm
#3001	[#_CLOCK1]	R/W	Clock 1 (ms)
#3002	[#_CLOCK2]	R/W	Clock 2 (hr)
#3003	[#_CNTL1]	R/W	Enable or disable the suppression of single block stop. Enable or disable the waiting of the auxiliary function completion signal.
#3003 bit 0	[#_M_SBK]	R/W	Enable or disable the suppression of single block stop.
#3003 bit 1	[#_M_FIN]	R/W	Enable or disable waiting for the auxiliary function completion signal.
#3004	[#_CNTL2]	R/W	Enable or disable feed hold. Enable or disable feedrate override. Enable or disable exact stop check.
#3004 bit 0	[#_M_FHD]	R/W	Enable or disable feed hold.
#3004 bit 1	[#_M_OV]	R/W	Enable or disable feedrate override.
#3004 bit 2	[#_M_EST]	R/W	Enable or disable exact stop check.
#3005	[#_SETDT]	R/W	Read/write setting data.
#3006	[#_MSGSTP]	W	Stop with a message.
#3007	[#_MRIMG]	R	Status of a mirror image (DI and setting)
#3008	[#_PRSTR]	R	Restarting/not restarting a program

- Time

System variable number	System variable name	Attribute	Description
#3011	[#_DATE]	R	Year/Month/Date
#3012	[#_TIME]	R	Hour/Minute/Second

- Path number of the parameter to be read or written

System variable number	System variable name	Attribute	Description
#3018	—	R/W	Path number of the parameter to be read or written

- Number of the path on which a macro is being executed

System variable number	System variable name	Attribute	Description
#3019	[#_PATH_NO]	R	Number of the path on which a macro is being executed

- Number of parts

System variable number	System variable name	Attribute	Description
#3901	[#_PRTSA]	R/W	Total number of parts
#3902	[#_PRTSN]	R/W	Number of required parts

- Tool compensation memory

M

System variable number	System variable name	Attribute	Description
#3980	[#_OFSMEM]	R	Tool compensation memory information

- Main program number

System variable number	System variable name	Attribute	Description
#4000	[#_MAINO]	R	Main program number

- Modal information

M

System variable number	System variable name	Attribute	Description
#4001-#4030	[#_BUFG[n]]	R	Modal information on blocks that have been specified by last minute (G code) Note) Subscript n represents a G code group number.
#4102	[#_BUFB]	R	Modal information on blocks that have been specified by last minute (B code)
#4107	[#_BUFD]	R	Modal information on blocks that have been specified by last minute (D code)
#4108	[#_BUFE]	R	Modal information on blocks that have been specified by last minute (E code)
#4109	[#_BUFF]	R	Modal information on blocks that have been specified by last minute (F code)
#4111	[#_BUFH]	R	Modal information on blocks that have been specified by last minute (H code)
#4113	[#_BUFM]	R	Modal information on blocks that have been specified by last minute (M code)

System variable number	System variable name	Attribute	Description
#4114	[#_BUFN]	R	Modal information on blocks that have been specified by last minute (sequence number)
#4115	[#_BUFO]	R	Modal information on blocks that have been specified by last minute (program number)
#4119	[#_BUFS]	R	Modal information on blocks that have been specified by last minute (S code)
#4120	[#_BUFT]	R	Modal information on blocks that have been specified by last minute (T code)
#4130	[#_BUFWZP]	R	Modal information on blocks that have been specified by last minute (additional workpiece coordinate system number)
#4201-#4230	[#_ACTG[n]]	R	Modal information on the block currently being executed (G code) Note) Subscript n represents a G code group number.
#4302	[#_ACTB]	R	Modal information on the block currently being executed (B code)
#4307	[#_ACTD]	R	Modal information on the block currently being executed (D code)
#4308	[#_ACTE]	R	Modal information on the block currently being executed (E code)
#4309	[#_ACTF]	R	Modal information on the block currently being executed (F code)
#4311	[#_ACTH]	R	Modal information on the block currently being executed (H code)
#4313	[#_ACTM]	R	Modal information on the block currently being executed (M code)
#4314	[#_ACTN]	R	Modal information on the block currently being executed (sequence number)
#4315	[#_ACTO]	R	Modal information on the block currently being executed (program number)
#4319	[#_ACTS]	R	Modal information on the block currently being executed (S code)
#4320	[#_ACTT]	R	Modal information on the block currently being executed (T code)
#4330	[#_ACTWZP]	R	Modal information on the block currently being executed (additional workpiece coordinate system number)
#4401-#4430	[#_INTG[n]]	R	Modal information on interrupted blocks (G code) Note) Subscript n represents a G code group number.
#4502	[#_INTB]	R	Modal information on interrupted blocks (B code)
#4507	[#_INTD]	R	Modal information on interrupted blocks (D code)
#4508	[#_INTE]	R	Modal information on interrupted blocks (E code)
#4509	[#_INTF]	R	Modal information on interrupted blocks (F code)
#4511	[#_INTH]	R	Modal information on interrupted blocks (H code)
#4513	[#_INTM]	R	Modal information on interrupted blocks (M code)
#4514	[#_INTN]	R	Modal information on interrupted blocks (sequence number)
#4515	[#_INTO]	R	Modal information on interrupted blocks (program number)
#4519	[#_INTS]	R	Modal information on interrupted blocks (S code)
#4520	[#_INTT]	R	Modal information on interrupted blocks (T code)
#4530	[#_INTWZP]	R	Modal information on interrupted blocks (additional workpiece coordinate system number)

T

System variable number	System variable name	Attribute	Description
#4001-#4030	[#_BUFG[n]]	R	Modal information on blocks that have been specified by last minute (G code) Note) Subscript n represents a G code group number.
#4108	[#_BUFE]	R	Modal information on blocks that have been specified by last minute (E code)
#4109	[#_BUFF]	R	Modal information on blocks that have been specified by last minute (F code)
#4113	[#_BUFM]	R	Modal information on blocks that have been specified by last minute (M code)
#4114	[#_BUFN]	R	Modal information on blocks that have been specified by last minute (sequence number)
#4115	[#_BUFO]	R	Modal information on blocks that have been specified by last minute (program number)
#4119	[#_BUFS]	R	Modal information on blocks that have been specified by last minute (S code)
#4120	[#_BUFT]	R	Modal information on blocks that have been specified by last minute (T code)
#4130	[#_BUFWZP]	R	Modal information on blocks that have been specified by last minute (additional workpiece coordinate system number)
#4201-#4230	[#_ACTG[n]]	R	Modal information on the block currently being executed (G code) Note) Subscript n represents a G code group number.
#4308	[#_ACTE]	R	Modal information on the block currently being executed (E code)
#4309	[#_ACTF]	R	Modal information on the block currently being executed (F code)
#4313	[#_ACTM]	R	Modal information on the block currently being executed (M code)
#4314	[#_ACTN]	R	Modal information on the block currently being executed (sequence number)
#4315	[#_ACTO]	R	Modal information on the block currently being executed (program number)
#4319	[#_ACTS]	R	Modal information on the block currently being executed (S code)
#4320	[#_ACTT]	R	Modal information on the block currently being executed (T code)
#4330	[#_ACTWZP]	R	Modal information on the block currently being executed (additional workpiece coordinate system number)
#4401-#4430	[#_INTG[n]]	R	Modal information on interrupted blocks (G code) Note) Subscript n represents a G code group number.
#4508	[#_INTE]	R	Modal information on interrupted blocks (E code)
#4509	[#_INTF]	R	Modal information on interrupted blocks (F code)
#4513	[#_INTM]	R	Modal information on interrupted blocks (M code)
#4514	[#_INTN]	R	Modal information on interrupted blocks (sequence number)
#4515	[#_INTO]	R	Modal information on interrupted blocks (program number)
#4519	[#_INTS]	R	Modal information on interrupted blocks (S code)
#4520	[#_INTT]	R	Modal information on interrupted blocks (T code)
#4530	[#_INTWZP]	R	Modal information on interrupted blocks (additional workpiece coordinate system number)

- Position information

System variable number	System variable name	Attribute	Description
#5001-#5020	[#_ABSIO[n]]	R	End point position of the previous block (workpiece coordinate system) Note) Subscript n represents an axis number (1 to 20)
#100001-#100050			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).
#5021-#5040	[#_ABSMT[n]]	R	Specified current position (machine coordinate system) Note) Subscript n represents an axis number (1 to 20).
#100051-#100100			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).
#5041-#5060	[#_ABSOT[n]]	R	Specified current position (workpiece coordinate system) Note) Subscript n represents an axis number (1 to 20).
#100101-#100150			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).
#5061-#5080	[#_ABSKP[n]]	R	Skip position (workpiece coordinate system) Note) Subscript n represents an axis number (1 to 20).
#100151-#100200			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).

- Tool length compensation value

M

System variable number	System variable name	Attribute	Description
#5081-#5100	[#_TOFS[n]]	R	Tool length compensation value Note) Subscript n represents an axis number (1 to 20).
#100201-#100250			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).

- Tool offset value

T

System variable number	System variable name	Attribute	Description
#5081, #100201	[#_TOFSWX]	R	X-axis tool offset (wear)
#5082, #100202	[#_TOFSWZ]		Z-axis tool offset (wear)
#5083, #100203	[#_TOFSWY]		Y-axis tool offset (wear)
#5084 - #5100	[#_TOFS[n]]		Tool offset (wear) for an arbitrary axis Note) Subscript n represents an axis number (4 to 20).
#100204-#100250			Tool offset (wear) for an arbitrary axis Note) Subscript n represents an axis number (4 to 50).
#5121, #100901	[#_TOFSGX]	R	X-axis tool offset (geometry)
#5122, #100902	[#_TOFSGZ]		Y-axis tool offset (geometry)
#5123, #100903	[#_TOFSGY]		Z-axis tool offset (geometry)
#5124 - #5140	[#_TOFSG[n]]		Tool offset (geometry) for an arbitrary axis Note) Subscript n represents an axis number (4 to 20).
#100904-#100950			Tool offset (geometry) for an arbitrary axis Note) Subscript n represents an axis number (4 to 50).

X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- Servo position deviation

System variable number	System variable name	Attribute	Description
#5101-#5120	[#_SVERR[n]]	R	Servo positional deviation Note) Subscript n represents an axis number (1 to 20).
#100251-#100300			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).

- Manual handle interruption

System variable number	System variable name	Attribute	Description
#5121-#5140	[#_MIRTP[n]]	R	Manual handle interruption Note) Subscript n represents an axis number (1 to 20).
#100651-#100700			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).

- Distance to go

System variable number	System variable name	Attribute	Description
#5181-#5200	[#_DIST[n]]	R	Distance to go Note) Subscript n represents an axis number (1 to 20).
#100801-#100850			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).

- Workpiece origin offset value, extended workpiece origin offset value

M

System variable number	System variable name	Attribute	Description
#5201-#5220	[#_WZCMN[n]]	R/W	External workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5221-#5240	[#_WZG54[n]]	R/W	G54 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5241-#5260	[#_WZG55[n]]	R/W	G55 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5261-#5280	[#_WZG56[n]]	R/W	G56 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5281-#5300	[#_WZG57[n]]	R/W	G57 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5301-#5320	[#_WZG58[n]]	R/W	G58 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5321-#5340	[#_WZG59[n]]	R/W	G59 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#100301-#100350	[#_WZCMN[n]]	R/W	External workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100351-#100400	[#_WZG54[n]]	R/W	G54 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100401-#100450	[#_WZG55[n]]	R/W	G55 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100451-#100500	[#_WZG56[n]]	R/W	G56 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100501-#100550	[#_WZG57[n]]	R/W	G57 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100551-#100600	[#_WZG58[n]]	R/W	G58 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).

System variable number	System variable name	Attribute	Description
#100601-#100650	[#_WZG59[n]]	R/W	G59 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
Extended workpiece origin offset value			
#7001-#7020	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#7021-#7040	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
:	:	:	:
:	:	:	:
#7941-#7960	[#_WZP48[n]]	R/W	G54.1P48 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#14001-#14020	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#14021-#14040	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
:	:	:	:
:	:	:	:
#19981-#20000	[#_WZP300[n]]	R/W	G54.1P300 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#101001-#101050	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#101051-#101100	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
:	:	:	:
:	:	:	:
#115901-#115950	[#_WZP299[n]]	R/W	G54.1P299 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#115951-#116000	[#_WZP300[n]]	R/W	G54.1P300 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).

T

System variable number	System variable name	Attribute	Description
#5201-#5220	[#_WZCMN[n]]	R/W	External workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5221-#5240	[#_WZG54[n]]	R/W	G54 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5241-#5260	[#_WZG55[n]]	R/W	G55 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5261-#5280	[#_WZG56[n]]	R/W	G56 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5281-#5300	[#_WZG57[n]]	R/W	G57 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5301-#5320	[#_WZG58[n]]	R/W	G58 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#5321-#5340	[#_WZG59[n]]	R/W	G59 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#100301-#100350	[#_WZCMN[n]]	R/W	External workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100351-#100400	[#_WZG54[n]]	R/W	G54 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100401-#100450	[#_WZG55[n]]	R/W	G55 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).

System variable number	System variable name	Attribute	Description
#100451-#100500	[#_WZG56[n]]	R/W	G56 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100501-#100550	[#_WZG57[n]]	R/W	G57 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100551-#100600	[#_WZG58[n]]	R/W	G58 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#100601-#100650	[#_WZG59[n]]	R/W	G59 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
Extended workpiece origin offset value			
#7001-#7020	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#7021-#7040	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
: :	: :	: :	: :
#7941-#7960	[#_WZP48[n]]	R/W	G54.1P48 workpiece origin offset value Note) Subscript n represents an axis number (1 to 20).
#101001-#101050	[#_WZP1[n]]	R/W	G54.1P1 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#101051-#101100	[#_WZP2[n]]	R/W	G54.1P2 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
: :	: :	: :	: :
#115901-#115950	[#_WZP299[n]]	R/W	G54.1P299 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).
#115951-#116000	[#_WZP300[n]]	R/W	G54.1P300 workpiece origin offset value Note) Subscript n represents an axis number (1 to 50).

- Skip position (detection unit)

System variable number	System variable name	Attribute	Description
#5421-#5440	[#_SKPDTC[n]]	R	Skip position (detection unit) Note) Subscript n represents an axis number (1 to 20).
#100701-#100750			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).

- Rotary table dynamic fixture offset value

M

System variable number	System variable name	Attribute	Description
#5500	[#_FOFSP]	R	Number of the standard fixture offset being selected (P)
#5501-#5520	[#_FOFSVAL[n]]	R	Offset value of the standard fixture offset being selected Note) Subscript n represents an axis number (1 to 20).
#117001-#117050			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).
#5521-#5540	[#_FOFS1[n]]	R/W	Standard fixture offset value (first set) Note) Subscript n represents an axis number (1 to 20).
#117051-#117100			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).
#5541-#5560	[#_FOFS2[n]]	R/W	Standard fixture offset value (second set) Note) Subscript n represents an axis number (1 to 20).
#117101-#117150			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).

System variable number	System variable name	Attribute	Description
#5561-#5580	[#_FOFS3[n]]	R/W	Standard fixture offset value (third set) Note) Subscript n represents an axis number (1 to 20).
#117151-#117200			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).
#5581-#5600	[#_FOFS4[n]]	R/W	Standard fixture offset value (fourth set) Note) Subscript n represents an axis number (1 to 20).
#117201-#117250			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).
#5601-#5620	[#_FOFS5[n]]	R/W	Standard fixture offset value (fifth set) Note) Subscript n represents an axis number (1 to 20).
#117251-#117300			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).
#5621-#5640	[#_FOFS6[n]]	R/W	Standard fixture offset value (sixth set) Note) Subscript n represents an axis number (1 to 20).
#117301-#117350			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).
#5641-#5660	[#_FOFS7[n]]	R/W	Standard fixture offset value (seventh set) Note) Subscript n represents an axis number (1 to 20).
#117351-#117400			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).
#5661-#5680	[#_FOFS8[n]]	R/W	Standard fixture offset value (eighth set) Note) Subscript n represents an axis number (1 to 20).
#117401-#117450			The numbers to the left can also be used. Note) Subscript n represents an axis number (1 to 50).

- Second geometry tool offset value

T

System variable number	System variable name	Attribute	Description
#5801-#5832 #27001-#27999	[#_OFSX2G[n]]	R/W	Second geometry tool offset X-axis compensation value Note) Subscript n represents a compensation number (1 to 32). If the number of pairs is greater than 32, the numbers on the left are also permitted. Note) Subscript n represents a compensation number (1 to 999).
#5833-#5864 #28001-#28999	[#_OFSZ2G[n]]	R/W	Second geometry tool offset Z-axis compensation value Note) Subscript n represents a compensation number (1 to 32). If the number of pairs is greater than 32, the numbers on the left are also permitted. Note) Subscript n represents a compensation number (1 to 999).
#5865-#5896 #29001-#29999	[#_OFSY2G[n]]	R/W	Second geometry tool offset Y-axis compensation value Note) Subscript n represents a compensation number (1 to 32). If the number of pairs is greater than 32, the numbers on the left are also permitted. Note) Subscript n represents a compensation number (1 to 999).

- Feedrate reduction ratio for rapid traverse overlap

System variable number	System variable name	Attribute	Description
#100851- #100900	[#_ROVLP [n]]	R/W	Feedrate reduction ratio for rapid traverse overlap Note) Subscript n represents an axis number (1 to 50).

- Serial spindle

System variable number	System variable name	Attribute	Description
#100951-#100954	[#_SPSTAT[n]]	R	State of each spindle Note) Subscript n represents a spindle number (1 to 4).

- Dynamic standard tool compensation value

M

System variable number	System variable name	Attribute	Description
#118051-#118100	[#_DOFS1[n]]	R/W	Dynamic standard tool compensation value (first set) Note) Subscript n represents an axis number (1 to 50).
#118101-#118150	[#_DOFS2[n]]	R/W	Dynamic standard tool compensation value (second set) Note) Subscript n represents an axis number (1 to 50).
#118151-#118200	[#_DOFS3[n]]	R/W	Dynamic standard tool compensation value (third set) Note) Subscript n represents an axis number (1 to 50).
#118201-#118250	[#_DOFS4[n]]	R/W	Dynamic standard tool compensation value (fourth set) Note) Subscript n represents an axis number (1 to 50).
#118251-#118300	[#_DOFS5[n]]	R/W	Dynamic standard tool compensation value (fifth set) Note) Subscript n represents an axis number (1 to 50).
#118301-#118350	[#_DOFS6[n]]	R/W	Dynamic standard tool compensation value (sixth set) Note) Subscript n represents an axis number (1 to 50).
#118351-#118400	[#_DOFS7[n]]	R/W	Dynamic standard tool compensation value (seventh set) Note) Subscript n represents an axis number (1 to 50).
#118401-#118450	[#_DOFS8[n]]	R/W	Dynamic standard tool compensation value (eighth set) Note) Subscript n represents an axis number (1 to 50).

- Local coordinate system offset value

System variable number	System variable name	Attribute	Description
#118501-#118550	[#_LCLOFS[n]]	R	Local coordinate system offset value Note) Subscript n represents an axis number (1 to 50).

- Others

System constant number	System constant name	Attribute	Description
#8570	-	R/W	Switching of P-CODE variables/system variables (#10000 and above)

- System constant

System constant number	System constant name	Attribute	Description
#0, #3100	[#_EMPTY]	R	Null
#3101	[#_PI]	R	Circular constant π = 3.14159265358979323846
#3102	[#_E]	R	Base of natural logarithm e = 2.71828182845904523536

Explanation

R, W, and R/W are attributes of a variable and represents read-only, write-only, and read/write enabled, respectively.

- Interface signal #1000-#1031, #1032, #1033-#1035 (Attribute: R) #1100-#1115, #1132, #1133-#1135 (Attribute: R/W)

[Input signal]

The status of interface input signals can be obtained by reading the value of system variables #1000 to #1032.

Variable number	Variable name	Point	Interface input signal
#1000	[#_UI[0]]	1	UI000 (2^0)
#1001	[#_UI[1]]	1	UI001 (2^1)
#1002	[#_UI[2]]	1	UI002 (2^2)
#1003	[#_UI[3]]	1	UI003 (2^3)
#1004	[#_UI[4]]	1	UI004 (2^4)
#1005	[#_UI[5]]	1	UI005 (2^5)
#1006	[#_UI[6]]	1	UI006 (2^6)
#1007	[#_UI[7]]	1	UI007 (2^7)
#1008	[#_UI[8]]	1	UI008 (2^8)
#1009	[#_UI[9]]	1	UI009 (2^9)
#1010	[#_UI[10]]	1	UI010 (2^{10})
#1011	[#_UI[11]]	1	UI011 (2^{11})
#1012	[#_UI[12]]	1	UI012 (2^{12})
#1013	[#_UI[13]]	1	UI013 (2^{13})
#1014	[#_UI[14]]	1	UI014 (2^{14})
#1015	[#_UI[15]]	1	UI015 (2^{15})
#1016	[#_UI[16]]	1	UI016 (2^{16})
#1017	[#_UI[17]]	1	UI017 (2^{17})
#1018	[#_UI[18]]	1	UI018 (2^{18})
#1019	[#_UI[19]]	1	UI019 (2^{19})
#1020	[#_UI[20]]	1	UI020 (2^{20})
#1021	[#_UI[21]]	1	UI021 (2^{21})
#1022	[#_UI[22]]	1	UI022 (2^{22})
#1023	[#_UI[23]]	1	UI023 (2^{23})
#1024	[#_UI[24]]	1	UI024 (2^{24})
#1025	[#_UI[25]]	1	UI025 (2^{25})
#1026	[#_UI[26]]	1	UI026 (2^{26})
#1027	[#_UI[27]]	1	UI027 (2^{27})
#1028	[#_UI[28]]	1	UI028 (2^{28})
#1029	[#_UI[29]]	1	UI029 (2^{29})
#1030	[#_UI[30]]	1	UI030 (2^{30})
#1031	[#_UI[31]]	1	UI031 (2^{31})
#1032	[#_UIL[0]]	32	UI000-UI031
#1033	[#_UIL[1]]	32	UI100-UI131
#1034	[#_UIL[2]]	32	UI200-UI231
#1035	[#_UIL[3]]	32	UI300-UI331

Variable value	Input signal
1.0	Contact closed
0.0	Contact opened

Since the read value is 1.0 or 0.0 regardless of the unit system, the unit system must be considered when a macro is created.

The input signals at 32 points can be read at a time by reading from system variables #1032 to #1035.

$$\#1032 = \sum_{i=0}^{30} \#[1000+i] \times 2^i - \#1031 \times 2^{31}$$

$$\#[1032+n] = \sum_{i=0}^{30} \left\{ 2^i \times V_i \right\} - 2^{31} \times V_{31}$$

When UIn_i = 0, V_i = 0.
When UIn_i = 1, V_i = 1.
n = 0-3

[Output signal]
Interface output signals can be sent by assigning values to system variables #1100 to #1132 for sending interface signals.

Variable number	Variable name	Point	Interface input signal
#1100	[#_UO[0]]	1	UO000 (2^0)
#1101	[#_UO[1]]	1	UO001 (2^1)
#1102	[#_UO[2]]	1	UO002 (2^2)
#1103	[#_UO[3]]	1	UO003 (2^3)
#1104	[#_UO[4]]	1	UO004 (2^4)
#1105	[#_UO[5]]	1	UO005 (2^5)
#1106	[#_UO[6]]	1	UO006 (2^6)
#1107	[#_UO[7]]	1	UO007 (2^7)
#1108	[#_UO[8]]	1	UO008 (2^8)
#1109	[#_UO[9]]	1	UO009 (2^9)
#1110	[#_UO[10]]	1	UO010 (2^{10})
#1111	[#_UO[11]]	1	UO011 (2^{11})
#1112	[#_UO[12]]	1	UO012 (2^{12})
#1113	[#_UO[13]]	1	UO013 (2^{13})
#1114	[#_UO[14]]	1	UO014 (2^{14})
#1115	[#_UO[15]]	1	UO015 (2^{15})
#1116	[#_UO[16]]	1	UO016 (2^{16})
#1117	[#_UO[17]]	1	UO017 (2^{17})
#1118	[#_UO[18]]	1	UO018 (2^{18})
#1119	[#_UO[19]]	1	UO019 (2^{19})
#1120	[#_UO[20]]	1	UO020 (2^{20})
#1121	[#_UO[21]]	1	UO021 (2^{21})
#1122	[#_UO[22]]	1	UO022 (2^{22})
#1123	[#_UO[23]]	1	UO023 (2^{23})
#1124	[#_UO[24]]	1	UO024 (2^{24})
#1125	[#_UO[25]]	1	UO025 (2^{25})
#1126	[#_UO[26]]	1	UO026 (2^{26})
#1127	[#_UO[27]]	1	UO027 (2^{27})
#1128	[#_UO[28]]	1	UO028 (2^{28})
#1129	[#_UO[29]]	1	UO029 (2^{29})
#1130	[#_UO[30]]	1	UO030 (2^{30})
#1131	[#_UO[31]]	1	UO031 (2^{31})
#1132	[#_UOL[0]]	32	UO000-UO031
#1133	[#_UOL[1]]	32	UO100-UO131
#1134	[#_UOL[2]]	32	UO200-UO231
#1135	[#_UOL[3]]	32	UO300-UO331

Variable value	Input signal
1.0	Contact closed

Variable value	Input signal
0.0	Contact opened

The output signals at 32 points can be written at a time by writing to system variables #1132 to #1135. The signals can also be read.

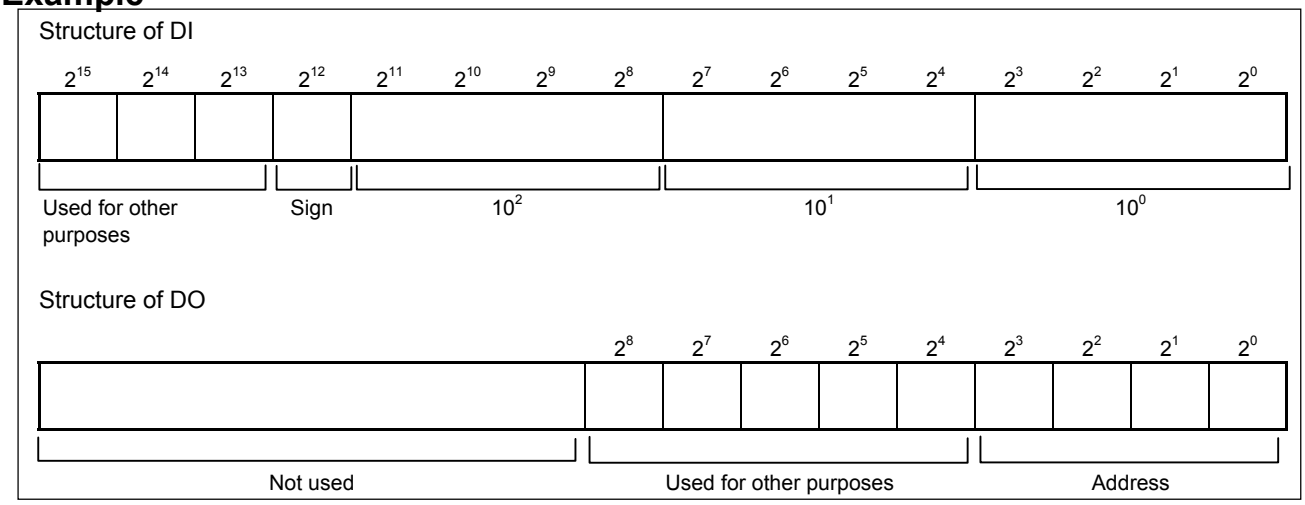
$$\#1132 = \sum_{i=0}^{30} \#[1000+i] \times 2^i - \#1131 \times 2^{31}$$

$$\#[1132+n] = \sum_{i=0}^{30} \left\{2^i \times V_i\right\} - 2^{31} \times V_{31}$$

When UIn_i = 0, V_i = 0.
When UIn_i = 1, V_i = 1.
n = 0-3

NOTE
1 When a value other than 1.0 or 0.0 is assigned to variables #1100 to #1131, it is assumed as follows.
<null> is assumed to be 0.
A value other than <null> or 0 is assumed to be 1.
Where, a value less than 0.00000001 is undefined.
2 When any of UI016 to UI031, UI100 to UI131, UI200 to UI231, UI300 to UI331, UO016 to UO031, UO200 to UO231, and UO300 to UO331 are used, bit 0 (MIF) of parameter No. 6001 must be set to 1.

Example



<1> The specified address D value is output to DO and signed BCD 3 digits are read into #100.

Macro calling instruction

```
G65 P9100 D (address);
```

A custom macro body is created as follows.

```
O9100 ;
#1132 = #1132 AND 496 OR#7 ;     : Address output
G65 P9101 T60 ;                  : Timer macro. For details, refer to the example of clock (#3001).
#100 = BIN[#1032 AND 4095] ;     : BCD 3 digits are read.
IF [#1012 EQ 0] GOTO 9100 ;      : A sign is attached.
#100 = -#100
N9100 M99
```

<2> Eight types of specified address D values are output to DO and signed BCD 6 digits (3-digit integer part + 3-digit fractional part) are read into #101.

Structure on the machine side

```
When DO 2^0 = 0:          Data with 3 decimal places
When DO 2^0 = 1:          Data with 3-digit integer part
When DO 2^3 to 2^1 = 000: No1 data when #1 = 0
When DO 2^3 to 2^1 = 001: No2 data when #2 = 0
      :
When DO 2^3 to 2^1 = 111: No8 data when #8 = 0
```

Macro calling instruction

```
G65    P9111    D (data number);
```

A custom macro body is created as follows.

```
O9111 ;
G65 P9100 D[#7*2+1] ;            : Call the macro of <1> O9100
#101 = #100   ;
G65 P9100 D[#7*2] ;
#101 = #101 + #100/1000 ;
M99 ;
```

- Interface signal R address #1036 to #1067, #1068, #1069 to #1071 (Attribute: R) #1136 to #1167, #1168, #1169 to #1171 (Attribute: R/W)

By setting bit 2 (IFR) of parameter No. 6020 to 1, this function is enabled. Set the start address of each R area in parameter No. 6093 or 6094. From the start address, 128 signal inputs or outputs are assigned. Set a multiple of 4 (0, 4, 8, ...) in parameters Nos. 6093 and 6094.

[Input signal]
The status of the input signal at each R address can be read with the value of system variable #1036 to #1071.

Variable number	Variable name	Point	Interface input signal
#1036	[#_UI[32]]	1	UI400 (2^0)
#1037	[#_UI[33]]	1	UI401 (2^1)
to	to	to	to
#1067	[#_UI[63]]	1	UI431 (2^{31})
#1068	[#_UIL[4]]	32	UI400 to UI431
#1069	[#_UIL[5]]	32	UI500 to UI531
#1070	[#_UIL[6]]	32	UI600 to UI631
#1071	[#_UIL[7]]	32	UI700 to UI731

Variable value	Input signal
1.0	Contact closed
0.0	Contact opened

[Output signal]
A value can be written to the output signal at each R address and the status of the signal can be read with the value of system variable #1136 to #1171.

Variable number	Variable name	Point	Interface input signal
#1136	[#_UO[32]]	1	UO400 (2^0)
#1137	[#_UO[33]]	1	UO401 (2^1)
to	to	to	to
#1167	[#_UO[63]]	1	UO431 (2^{31})
#1168	[#_UOL[4]]	32	UO400 to UO431
#1169	[#_UOL[5]]	32	UO500 to UO531
#1170	[#_UOL[6]]	32	UO600 to UO631
#1171	[#_UOL[7]]	32	UO700 to UO731

Variable value	Input signal
1.0	Contact closed
0.0	Contact opened

NOTE
1 When signals UO400 to UO431 are handled from system variables #1136 to #1171, do not write any of signals UO400 to UO431 from a ladder or another interface. Specifying a write may cause a conflict between read from and write to the signal, resulting in an unexpected change to the signal.
2 For signals UO400 to UO731, R addresses are used, while for signals UO000 to UO331, F addresses are used. Therefore, to handle these signals with a ladder, it is necessary to consider synchronization processing of I/O signals. For details, refer to Subsection 1.4.5, “Synchronization Processing of I/O Signals”, in the “PMC Programming Manual (B-64513EN)”.

- Tool compensation value #2001-#2800, #10001-#13999, #21001-#22999 (Attribute: R/W)

M

The compensation values can be obtained by reading system variables #2001 to #2800, #10001 to #13999, or #21001 to #22999 for tool compensation. The compensation values can also be changed by assigning values to the system variables.

<1> Tool compensation memory A

- When the number of compensations is 200 or less

Compensation number	Variable number	Variable name
1	#2001	[#_OFS[1]]
2	#2002	[#_OFS[2]]
3	#2003	[#_OFS[3]]
:	:	:
199	#2199	[#_OFS[199]]
200	#2200	[#_OFS[200]]

- When the number of compensations is more than 200 (For compensation with a compensation number of 200 or less, #2001 to #2200 can also be used.)

Compensation number	Variable number	Variable name
1	#10001	[#_OFS[1]]
2	#10002	[#_OFS[2]]
3	#10003	[#_OFS[3]]
:	:	:
998	#10998	[#_OFS[998]]
999	#10999	[#_OFS[999]]

<2> Tool compensation memory B

- When the number of compensations is 200 or less
  When bit 3 (V15) of parameter No.6000 = 0

Compensation number	Wear		Geometry	
	Variable number	Variable name	Variable number	Variable name
1	#2001	[#_OFSW[1]]	#2201	[#_OFSG[1]]
2	#2002	[#_OFSW[2]]	#2202	[#_OFSG[2]]
3	#2003	[#_OFSW[3]]	#2203	[#_OFSG[3]]
:	:	:	:	:
199	#2199	[#_OFSW[199]]	#2399	[#_OFSG[199]]
200	#2200	[#_OFSW[200]]	#2400	[#_OFSG[200]]

When bit 3 (V15) of parameter No.6000 = 1

Compensation number	Wear		Geometry	
	Variable number	Variable name	Variable number	Variable name
1	#2201	[#_OFSW[1]]	#2001	[#_OFSG[1]]
2	#2202	[#_OFSW[2]]	#2002	[#_OFSG[2]]
3	#2203	[#_OFSW[3]]	#2003	[#_OFSG[3]]
:	:	:	:	:
199	#2399	[#_OFSW[199]]	#2199	[#_OFSG[199]]
200	#2400	[#_OFSW[200]]	#2200	[#_OFSG[200]]

- When the number of compensations is more than 200 (For compensation with a compensation number of 200 or less, #2001 to #2400 can also be used.)
  When bit 3 (V15) of parameter No.6000 = 0

Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#11001	[#_OFSG[1]]	#10001	[#_OFSW[1]]
2	#11002	[#_OFSG[2]]	#10002	[#_OFSW[2]]
3	#11003	[#_OFSG[3]]	#10003	[#_OFSW[3]]
:	:	:	:	:
998	#11998	[#_OFSG[998]]	#10998	[#_OFSW[998]]
999	#11999	[#_OFSG[999]]	#10999	[#_OFSW[999]]

When bit 3 (V15) of parameter No.6000 = 1

Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#10001	[#_OFSG[1]]	#11001	[#_OFSW[1]]
2	#10002	[#_OFSG[2]]	#11002	[#_OFSW[2]]
3	#10003	[#_OFSG[3]]	#11003	[#_OFSW[3]]
:	:	:	:	:
998	#10998	[#_OFSG[998]]	#11998	[#_OFSW[998]]
999	#10999	[#_OFSG[999]]	#11999	[#_OFSW[999]]

<3> Tool offset memory C

- When the number of compensations is 200 or less
  When bit 3 (V15) of parameter No.6000 = 0

H code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#2201	[#_OFSHG[1]] or [#_OFSZG[1]]	#2001	[#_OFSHW[1]] or [#_OFSZW[1]]
2	#2202	[#_OFSHG[2]] or [#_OFSZG[2]]	#2002	[#_OFSHW[2]] or [#_OFSZW[2]]
3	#2203	[#_OFSHG[3]] or [#_OFSZG[3]]	#2003	[#_OFSHW[3]] or [#_OFSZW[3]]
:	:	:	:	:
199	#2399	[#_OFSHG[199]] or [#_OFSZG[199]]	#2199	[#_OFSHW[199]] or [#_OFSZW[199]]
200	#2400	[#_OFSHG[200]] or [#_OFSZG[200]]	#2200	[#_OFSHW[200]] or [#_OFSZW[200]]

When bit 3 (V15) of parameter No.6000 = 1

H code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#2001	[#_OFSHG[1]] or [#_OFSZG[1]]	#2201	[#_OFSHW[1]] or [#_OFSZW[1]]
2	#2002	[#_OFSHG[2]] or [#_OFSZG[2]]	#2202	[#_OFSHW[2]] or [#_OFSZW[2]]
3	#2003	[#_OFSHG[3]] or [#_OFSZG[3]]	#2203	[#_OFSHW[3]] or [#_OFSZW[3]]

H code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
:	:	:	:	:
199	#2199	[#_OFSHG[199]] or [#_OFSZG[199]]	#2399	[#_OFSHW[199]] or [#_OFSZW[199]]
200	#2200	[#_OFSHG[200]] or [#_OFSZG[200]]	#2400	[#_OFSHW[200]] or [#_OFSZW[200]]

D code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#2401	[#_OFSDG[1]] or [#_OFSRG[1]]	#2601	[#_OFSDW[1]] or [#_OFSRW[1]]
2	#2402	[#_OFSDG[2]] or [#_OFSRG[2]]	#2602	[#_OFSDW[2]] or [#_OFSRW[2]]
3	#2403	[#_OFSDG[3]] or [#_OFSRG[3]]	#2603	[#_OFSDW[3]] or [#_OFSRW[3]]
:	:	:	:	:
199	#2599	[#_OFSDG[199]] or [#_OFSRG[199]]	#2799	[#_OFSDW[199]] or [#_OFSRW[199]]
200	#2600	[#_OFSDG[200]] or [#_OFSRG[200]]	#2800	[#_OFSDW[200]] or [#_OFSRW200]]

NOTE
1 When #2401 to #2800 are used for reading or writing of D codes, bit 5 (D15) of parameter No.6004 must be set to 1.
2 When bit 5 (D15) of parameter No.6004 is set to 1, system variables #2500 to #2806 for workpiece origin offset cannot be used. Use system variables #5201 to #5324.

- When the number of compensations is more than 200 (For compensation with a compensation number of 200 or less, #2001 to #2800 can also be used.)
  When bit 3 (V15) of parameter No.6000 = 0

H code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#11001	[#_OFSHG[1]] or [#_OFSZG[1]]	#10001	[#_OFSHW[1]] or [#_OFSZW[1]]
2	#11002	[#_OFSHG[2]] or [#_OFSZG[2]]	#10002	[#_OFSHW[2]] or [#_OFSZW[2]]
3	#11003	[#_OFSHG[3]] or [#_OFSZG[3]]	#10003	[#_OFSHW[3]] or [#_OFSZW[3]]
:	:	:	:	:

H code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
998	#11998	[#_OFSHG[998]] or [#_OFSZG[998]]	#10998	[#_OFSHW[998]] or [#_OFSZW[998]]
999	#11999	[#_OFSHG[999]] or [#_OFSZG[999]]	#10999	[#_OFSHW[999]] or [#_OFSZW[999]]

D code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#13001	[#_OFSDG[1]] or [#_OFSRG[1]]	#12001	[#_OFSDW[1]] or [#_OFSRW[1]]
2	#13002	[#_OFSDG[2]] or [#_OFSRG[2]]	#12002	[#_OFSDW[2]] or [#_OFSRW[2]]
3	#13003	[#_OFSDG[3]] or [#_OFSRG[3]]	#12003	[#_OFSDW[3]] or [#_OFSRW[3]]
:	:	:	:	:
998	#13998	[#_OFSDG[998]] or [#_OFSRG[998]]	#12998	[#_OFSDW[998]] or [#_OFSRW[998]]
999	#13999	[#_OFSDG[999]] or [#_OFSRG[999]]	#12999	[#_OFSDW[999]] or [#_OFSRW[999]]

When bit 3 (V15) of parameter No.6000 = 1

H code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#10001	[#_OFSHG[1]] or [#_OFSZG[1]]	#11001	[#_OFSHW[1]] or [#_OFSZW[1]]
2	#10002	[#_OFSHG[2]] or [#_OFSZG[2]]	#11002	[#_OFSHW[2]] or [#_OFSZW[2]]
3	#10003	[#_OFSHG[3]] or [#_OFSZG[3]]	#11003	[#_OFSHW[3]] or [#_OFSZW[3]]
:	:	:	:	:
998	#10998	[#_OFSHG[998]] or [#_OFSZG[998]]	#11998	[#_OFSHW[998]] or [#_OFSZW[998]]
999	#10999	[#_OFSHG[999]] or [#_OFSZG[999]]	#11999	[#_OFSHW[999]] or [#_OFSZW[999]]

D code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#12001	[#_OFSDG[1]] or [#_OFSRG[1]]	#13001	[#_OFSDW[1]] or [#_OFSRW[1]]
2	#12002	[#_OFSDG[2]] or [#_OFSRG[2]]	#13002	[#_OFSDW[2]] or [#_OFSRW[2]]
3	#12003	[#_OFSDG[3]] or [#_OFSRG[3]]	#13003	[#_OFSDW[3]] or [#_OFSRW[3]]
:	:	:	:	:
998	#12998	[#_OFSDG[998]] or [#_OFSRG[998]]	#13998	[#_OFSDW[998]] or [#_OFSRW[998]]
999	#12999	[#_OFSDG[999]] or [#_OFSRG[999]]	#13999	[#_OFSDW[999]] or [#_OFSRW[999]]

System variables not dependent on bit 3 (V15) of parameter No. 6000

Corner R offset				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#21001	[#_CORR_G[1]]	#22001	[#_CORR_W[1]]
2	#21002	[#_CORR_G[2]]	#22002	[#_CORR_W[2]]
3	#21003	[#_CORR_G[3]]	#22003	[#_CORR_W[3]]
:	:	:	:	:
998	#21998	[#_CORR_G[998]]	#22998	[#_CORR_W[998]]
999	#21999	[#_CORR_G[999]]	#22999	[#_CORR_W[999]]

- Tool compensation value #2001-#2964, #10001-#19999 (Attribute: R/W)

T

The compensation values can be obtained by reading system variables #2001 to #2964 or #10001 to #19999 for tool compensation. The compensation values can also be changed by assigning values to the system variables.

<1> Without tool geometry/wear compensation memory

- When the number of compensations is 64 or less

Compensation number	Variable number	Variable name	Description
1	#2001	[#_OFSX[1]]	X-axis compensation value (*1)
2	#2002	[#_OFSX[2]]	
3	#2003	[#_OFSX[3]]	
:	:	:	
63	#2063	[#_OFSX[63]]	
64	#2064	[#_OFSX[64]]	
1	#2101	[#_OFSZ[1]]	Z-axis compensation value (*1)
2	#2102	[#_OFSZ[2]]	
3	#2103	[#_OFSZ[3]]	
:	:	:	
63	#2163	[#_OFSZ[63]]	
64	#2164	[#_OFSZ[64]]	

Compensation number	Variable number	Variable name	Description
1	#2201	[#_OFSR[1]]	Tool nose radius compensation value
2	#2202	[#_OFSR[2]]	
3	#2203	[#_OFSR[3]]	
:	:	:	
63	#2263	[#_OFSR[63]]	
64	#2264	[#_OFSR[64]]	
1	#2301	[#_OFST[1]]	Virtual tool tipT position
2	#2302	[#_OFST[2]]	
3	#2303	[#_OFST[3]]	
:	:	:	
63	#2363	[#_OFST[63]]	
64	#2364	[#_OFST[64]]	
1	#2401	[#_OFSY[1]]	Y-axis compensation value (*1)
2	#2402	[#_OFSY[2]]	
3	#2403	[#_OFSY[3]]	
:	:	:	
48	#2448	[#_OFSY[48]]	
49	#2449	[#_OFSY[49]]	

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- When the number of compensations is more than 64 (For compensation with a compensation number of 64 or less, #2001 to #2449 can also be used.)

Compensation number	Variable number	Variable name	Description
1	#10001	[#_OFSX[1]]	X-axis compensation value (*1)
2	#10002	[#_OFSX[2]]	
3	#10003	[#_OFSX[3]]	
:	:	:	
998	#10998	[#_OFSX[998]]	
999	#10999	[#_OFSX[999]]	
1	#11001	[#_OFSZ[1]]	Z-axis compensation value (*1)
2	#11002	[#_OFSZ[2]]	
3	#11003	[#_OFSZ[3]]	
:	:	:	
998	#11998	[#_OFSZ[998]]	
999	#11999	[#_OFSZ[999]]	
1	#12001	[#_OFSR[1]]	Tool nose radius compensation value
2	#12002	[#_OFSR[2]]	
3	#12003	[#_OFSR[3]]	
:	:	:	
998	#12998	[#_OFSR[998]]	
999	#12999	[#_OFSR[999]]	
1	#13001	[#_OFST[1]]	Virtual tool tip T position
2	#13002	[#_OFST[2]]	
3	#13003	[#_OFST[3]]	
:	:	:	
998	#13998	[#_OFST[998]]	
999	#13999	[#_OFST[999]]	
1	#14001	[#_OFSY[1]]	Y-axis compensation value (*1)
2	#14002	[#_OFSY[2]]	
3	#14003	[#_OFSY[3]]	
:	:	:	
998	#14998	[#_OFSY[998]]	
999	#14999	[#_OFSY[999]]	

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

<2> With tool geometry/wear compensation memory

- When the number of compensations is 64 or less

Compensation number	Variable number	Variable name	Description
1	#2001	[#_OFSXW[1]]	X-axis compensation value (wear) (*1)
2	#2002	[#_OFSXW[2]]	
3	#2003	[#_OFSXW[3]]	
:	:	:	
63	#2063	[#_OFSXW[63]]	
64	#2064	[#_OFSXW[64]]	
1	#2101	[#_OFSZW[1]]	Z-axis compensation value (wear) (*1)
2	#2102	[#_OFSZW[2]]	
3	#2103	[#_OFSZW[3]]	
:	:	:	
63	#2163	[#_OFSZW[63]]	
64	#2164	[#_OFSZW[64]]	
1	#2201	[#_OFSRW[1]]	Tool nose radius compensation value (wear)
2	#2202	[#_OFSRW [2]]	
3	#2203	[#_OFSRW [3]]	
:	:	:	
63	#2263	[#_OFSRW [63]]	
64	#2264	[#_OFSRW [64]]	
1	#2301	[#_OFST[1]]	Virtual tool tip T position
2	#2302	[#_OFST[2]]	
3	#2303	[#_OFST[3]]	
:	:	:	
63	#2363	[#_OFST[63]]	
64	#2364	[#_OFST[64]]	
1	#2401	[#_OFSYW[1]]	Y-axis compensation value (wear) (*1)
2	#2402	[#_OFSYW [2]]	
3	#2403	[#_OFSYW [3]]	
:	:	:	
48	#2448	[#_OFSYW [48]]	
49	#2449	[#_OFSYW [49]]	
1	#2451	[#_OFSYG[1]]	Y-axis compensation value (geometry) (*1)
2	#2452	[#_OFSYG [2]]	
3	#2453	[#_OFSYG [3]]	
:	:	:	
48	#2498	[#_OFSYG [48]]	
49	#2499	[#_OFSYG [49]]	
1	#2701	[#_OFSXG[1]]	X-axis compensation value (geometry) (*1)
2	#2702	[#_OFSXG[2]]	
3	#2703	[#_OFSXG [3]]	
:	:	:	
48	#2748	[#_OFSXG [48]]	
49	#2749	[#_OFSXG [49]]	
1	#2801	[#_OFSZG[1]]	Z-axis compensation value (geometry) (*1)
2	#2802	[#_OFSZG[2]]	
3	#2803	[#_OFSZG[3]]	
:	:	:	
48	#2848	[#_OFSZG[48]]	
49	#2849	[#_OFSZG[49]]	

Compensation number	Variable number	Variable name	Description
1	#2901	[#_OFSRG[1]]	Tool nose radius compensation value (geometry)
2	#2902	[#_OFSRG[2]]	
3	#2903	[#_OFSRG[3]]	
:	:	:	
63	#2963	[#_OFSRG[63]]	
64	#2964	[#_OFSRG[64]]	
1	#19001	[#_OFSYG[1]]	Y-axis compensation value (geometry) (*1)
2	#19002	[#_OFSYG[2]]	
3	#19003	[#_OFSYG[3]]	
:	:	:	
998	#19998	[#_OFSYG[998]]	
999	#19999	[#_OFSYG[999]]	

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- When the number of compensations is more than 64 (For compensation with a compensation number of 64 or less, #2001 to #2964 or #10001 to #19999 can also be used.)

Compensation number	Variable number	Variable name	Description
1	#2201	[#_OFSRW[1]]	X-axis compensation value (wear) (*1)
2	#2202	[#_OFSRW [2]]	
3	#2203	[#_OFSRW [3]]	
:	:	:	
63	#2263	[#_OFSRW [63]]	
64	#2264	[#_OFSRW [64]]	
1	#2301	[#_OFST[1]]	Z-axis compensation value (wear) (*1)
2	#2302	[#_OFST[2]]	
3	#2303	[#_OFST[3]]	
:	:	:	
63	#2363	[#_OFST[63]]	
64	#2364	[#_OFST[64]]	
1	#2401	[#_OFSYW[1]]	Tool nose radius compensation value (wear)
2	#2402	[#_OFSYW [2]]	
3	#2403	[#_OFSYW [3]]	
:	:	:	
48	#2448	[#_OFSYW [48]]	
49	#2449	[#_OFSYW [49]]	
1	#2451	[#_OFSYG[1]]	Virtual tool tip T position
2	#2452	[#_OFSYG [2]]	
3	#2453	[#_OFSYG [3]]	
:	:	:	
48	#2498	[#_OFSYG [48]]	
49	#2499	[#_OFSYG [49]]	
1	#2701	[#_OFSXG[1]]	Y-axis compensation value (wear) (*1)
2	#2702	[#_OFSXG[2]]	
3	#2703	[#_OFSXG [3]]	
:	:	:	
48	#2748	[#_OFSXG [48]]	
49	#2749	[#_OFSXG [49]]	

Compensation number	Variable number	Variable name	Description
1	#2901	[#_OFSRG[1]]	X-axis compensation value (geometry) (*1)
2	#2902	[#_OFSRG[2]]	
3	#2903	[#_OFSRG[3]]	
:	:	:	
63	#2963	[#_OFSRG[63]]	
64	#2964	[#_OFSRG[64]]	
1	#19001	[#_OFSYG[1]]	Z-axis compensation value (geometry) (*1)
2	#19002	[#_OFSYG[2]]	
3	#19003	[#_OFSYG[3]]	
:	:	:	
998	#19998	[#_OFSYG[998]]	
999	#19999	[#_OFSYG[999]]	
1	#2901	[#_OFSRG[1]]	Tool nose radius compensation value (geometry)
2	#2902	[#_OFSRG[2]]	
3	#2903	[#_OFSRG[3]]	
:	:	:	
63	#2963	[#_OFSRG[63]]	
64	#2964	[#_OFSRG[64]]	
1	#19001	[#_OFSYG[1]]	Y-axis compensation value (geometry) (*1)
2	#19002	[#_OFSYG[2]]	
3	#19003	[#_OFSYG[3]]	
:	:	:	
998	#19998	[#_OFSYG[998]]	
999	#19999	[#_OFSYG[999]]	

(*1) X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes

- Alarm #3000 (Attribute: W)

When an error is detected in a macro, an unit can enter the alarm state. In addition, an alarm message of up to 60 characters with alphabet and numerals can be specified between a control-out and a control-in after the expression. When an alarm message is not specified, a macro alarm is used instead.

Variable number	Variable name	Description
#3000	[#_ALM]	Macro alarm

When bit 1 (MCA) of parameter No.6008 = 0

#3000 = n (ALARM MESSAGE); (n: 0-200)

On the screen, the alarm number obtained by adding the value of #3000 to 3000 and alarm message appear after MC.

(Example) #3000 = 1 (ALARM MESSAGE);

=> "MC3001 ALARM MESSAGE" appears on the alarm screen.

When bit 1 (MCA) of parameter No.6008 = 1

#3000 = n (ALARM MESSAGE); (n: 0-4095)

On the screen, the alarm number of #3000 and alarm message appear after MC.

(Example) #3000=1 (ALARM MESSAGE);

=> "MC0001 ALARM MESSAGE" appears on the alarm screen.

When #3000 is commanded with the comment section, the order of alarm message and the comment section changes places by setting value of the parameter NCM(No.6020#3).

When parameter NCM(No.6020#3) is set to 0

It is assumed that it alarm message first, and assumes since the second to be comment section.

(Example) #3000 =1 (ALARM MESSAGE) (COMMENT 1) (COMMENT 2);

When parameter NCM(No.6020#3) is set to 1

The end is assumed to be alarm message, and it assumes to be comment section from it ahead.

(Example) #3000 =1 (COMMENT 1) (COMMENT 2) (ALARM MESSAGE);

- Clock #3001, #3002 (Attribute: R/W)

The clock time can be obtained by reading system variables #3001 and #3002 for clocks. The time can be preset by entering a value in the system variables.

Type	Variable number	Variable name	Unit	At power-on	Count condition
Clock 1	#3001	[#_CLOCK1]	1 ms	Reset to 0	Anytime
Clock 2	#3002	[#_CLOCK2]	1 hour	Same as at power-down	When the STL signal is on

The clock accuracy is 16 ms. Clock 1 returns to 0 after a lapse of 2147483648 ms. Clock 2 returns to 0 after a lapse of 9544.37176 hours.

[Example]

Timer

Macro calling command

G65 P9101 T (wait time) ms ;

A macro is created as follows.

```
O9101;
#3001 = 0;    Initial setting
WHILE [#3001 LE #20] DO1:    Wait for a specified time
END1 ;
M99 ;
```

- Controlling of single block stop and waiting for the auxiliary function completion signal #3003 (Attribute: R/W)

Assigning the following values in system variable #3003 allows the specification of whether single block stop is disabled in the following blocks or whether a wait for the completion signal (FIN) of the auxiliary function (M, S, T, or B) before going to the next block is enabled. When a wait for completion signal is disabled, the distribution end signal (DEN) is not sent. Be careful not to specify the next auxiliary function without waiting for the completion signal.

Variable number and variable name	Value	Single block stop	Auxiliary function completion signal
#3003 [#_CNTL1]	0	Enabled	Waiting
	1	Disabled	Waiting
	2	Enabled	Not waiting
	3	Disabled	Not waiting

In addition, the following variable names can be used to enable or disable single block stop and a wait for the auxiliary function completion signal, individually.

Variable name	Value	Single block stop	Auxiliary function completion
[#_M_SBK]	0	Enabled	-
	1	Disabled	-
[#_M_FIN]	0	-	Waiting
	1	-	Not waiting

[Example]

Drill cycle (for incremental programming)

(G81 equivalent)

Macro calling command

G65 P9081 L Iterations R R point Z Z point;

A custom macro body is created as follows.

```
O9081 ;
#3003 = 1 ;
G00 Z#18 ;
G01 Z#26 ;
G00 Z-[ ROUND[#18] + ROUND[#26] ] ;
#3003 = 0 ;
M99 ;
```

Disable single block stop.
#18 corresponds to R and #26 to Z.

NOTE
#3003 is cleared by a reset.

- Enabling of feed hold, feedrate override, and exact stop check #3004 (Attribute: R/W)

Assigning the following values in system variable #3004 allows the specification of whether feed hold and feedrate override are enabled in the following blocks or whether exact stop in G61 mode or by G09 command is disabled.

Variable number and variable name	Value	Field hold	Feedrate override	Exact stop
#3004 [#_CNTL2]	0	Enabled	Enabled	Enabled
	1	Disabled	Enabled	Enabled
	2	Enabled	Disabled	Enabled
	3	Disabled	Disabled	Enabled
	4	Enabled	Enabled	Disabled
	5	Disabled	Enabled	Disabled
	6	Enabled	Disabled	Disabled
	7	Disabled	Disabled	Disabled

In addition, the following variable names can be used to enable or disable feed hold, feedrate override, and exact stop in G61 mode or by the G09 command, individually.

Variable number and variable name	Value	Feed hold	Feedrate override	Exact stop
[#_M_FHD]	0	Enabled	-	-
	1	Disabled	-	-
[#_M_OV]	0	-	Enabled	-
	1	-	Disabled	-
[#_M_EST]	0	-	-	Enabled
	1	-	-	Disabled

NOTE
1 These system variables are provided to maintain compatibility with conventional NC programs. It is recommended that functions provided by G63, G09, G61, and other G codes be used to enable or disable feed hold, feedrate override, and exact stop.
2 When the feed hold button is pressed during execution of a block for which feed hold is disabled:
  <1> If the feed hold button is kept pressed, operation stops after execution of the block. When single block stop is disabled, however, operation does not stop.
  <2> If the pressed feed hold button is released, the feed hold lamp lights, but operation does not stop until the end of the first block that was enabled.
3 #3004 is cleared by a reset.

NOTE
4 If exact stop is disabled by #3004, the original exact stop position between cutting feed and positioning block is not affected. #3004 can temporarily disable exact stop in G61 mode or by the G09 command between cutting feed and cutting feed.

- Settings #3005 (Attribute: R/W)

Settings can be read and written.
Binary values are converted to decimals.

#3005

	#15	#14	#13	#12	#11	#10	#9	#8
Setting							FCV	

	#7	#6	#5	#4	#3	#2	#1	#0
Setting			SEQ			INI	ISO	TVC

#9 (FCV) : Whether to use the FANUC Series 15 program format conversion capability
#5 (SEQ) : Whether to automatically insert sequence numbers
#2 (INI) : Millimeter input or inch input
#1 (ISO) : Whether to use EIA or ISO as the output code
#0 (TVC) : Whether to make a TV check

- Stop with a message #3006 (Attribute: W)

When "#3006=1 (MESSAGE);" is commanded in the macro, the program executes blocks up to the immediately previous one and then stops. When a message of up to 60 characters with alphabet and numerals, which is enclosed by a control-in character and control-out character, is programmed in the same block, the message is displayed on the external operator message screen.

Variable number	Variable name	Description
#3006	[#_MSGSTP]	Stop with a message

When #3006 is commanded with the comment section, the order of message and the comment section changes places by setting value of the parameter NCM(No.6020#3).
When parameter NCM(No.6020#3) is set to 0
It is assumed that it message first, and assumes since the second to be comment section.
(Example) #3000 =1 (MACRO MESSAGE) (COMMENT 1) (COMMENT 2);
When parameter NCM(No.6020#3) is set to 1
The end is assumed to be message, and it assumes to be comment section from it ahead.
(Example) #3000 =1 (COMMENT 1) (COMMENT 2) (MACRO MESSAGE);

- Status of a mirror image #3007 (Attribute: R)

The status of an mirror image (setting or DI) at that point in time can be obtained for each axis by reading #3007.

Variable number	Variable name	Description
#3007	[#_MRIMG]	Status of a mirror image

When the status is indicated in binary, each bit corresponds with an axis as follows.

Bit	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16
nth axis	32	31	30	29	28	27	26	25	24	23	22	21	20	19	18	17

Bit	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
nth axis	16	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1

For the 32 bits, 0 indicates that a mirror image is disabled and 1 indicates that a mirror image is enabled.
[Example] When #3007 is 3, a mirror image is enabled for the 1st and 2nd axes.

NOTE
1 The status of a programmable mirror image is not reflected on this variable.
2 When the mirror image function is set for the same axis by the mirror image signal and setting, the signal value and setting value are ORed and then output.
3 When mirror image signals for axes other than the controlled axes are turned on, they are not read into system variable #3007.

- Status during restart of a program #3008 (Attribute: R)

Whether a program is restarting can be determined by reading #3008.

Variable number	Variable name	Description
#3008	[#_PRSTR]	0: Program is not restarting. 1: Program is restarting.

- Time #3011, #3012 (Attribute: R)

Year/month/date and hour/minute/second can be obtained by reading system variables #3011 and #3012. This variable is read-only. To change year/month/date and hour/minute/second, use the timer screen.

[Example] May 20, 2004, PM 04:17:05
#3011 = 20040520
#3012 = 161705

- Path number of the parameter to be read or written #3018 (Attribute: R/W)

If a parameter for another path is to be read or written using parameter reading with the operation command PRM or using parameter writing with programmable parameter input (G10L52), this system variable is used to specify that path.
Relationships between specified values and path numbers are as follows:

#3018	Selected path
0	Local path
1	Path 1
2	Path 2
:	:
10	Path 10

Variable number	Variable name	Description
#3018	—	Path number of the parameter to be read or written

- Number of the path on which a macro is being executed #3019 (Attribute: R)

This system variable can be used to read the number of the path on which a macro is being executed.
By using this variable, it is possible to execute a single program differently for different paths.
The path number can be read as follows:

Path	#3019
Path 1	1
Path 2	2
Path 3	3
:	:
Path 10	10

Variable number	Variable name	Description
#3019	[#_PATH_NO]	Path number

- Total number of parts and the number of required parts #3901 and #3902 (Attribute: R/W)

The number of required parts and the number of machined parts can be displayed on the screen by using the operation time and part number displaying function. When the (total) number of machined parts reaches the number of required parts, a signal indicating the fact is sent to the machine (PMC side).
The system variables can be used to read or write the total number of parts and the number of required parts.

Variable number	Variable name	Description
#3901	[#_PRTSA]	Total number of parts
#3902	[#_PRTSN]	Number of required parts

- Type of tool compensation memory #3980 (Attribute: R)

M

System variable #3980 can be used to read the type of compensation memory.

Variable number	Variable name	Description
#3980	[#_OFSMEM]	Types of tool compensation memory 0: Tool compensation memory A 1: Tool compensation memory B 2: Tool compensation memory C

- Main program number #4000 (Attribute: R)

System variable #4000 can be used to read the main program number regardless of the level of a subprogram.

Variable number	Variable name	Description
#4000	[#_MAINO]	Main program number

NOTE
1 The main program number indicates the number of the program that is first started.
2 When an O number is specified by MDI during execution of the main program or when the second O number is specified in DNC mode, the value of #4000 changes to the specified O number. In addition, when no programs are registered or when no O numbers are specified in DNC mode, the value of #4000 changes to 0.

- Modal information #4001-#4130, #4201-#4330, #4401-#4530 (Attribute: R)

The modal information specified before the previous block of the macro statement that reads system variables #4001 to #4130 can be obtained in the block currently being looked ahead, by reading system variables #4001 to #4130.
The modal information of the block currently being executed can be obtained by reading system variables #4201 to #4330.
The modal information specified before the block interrupted by an interruption type custom macro by reading system variables #4401 to #4530.
The unit used when it was specified is applied.

M

(Category: <1> Previous block, <2> Running block, <3> Interrupted block)

Category	Variable number	Variable name	Description
<1>	#4001	[#_BUFG[1]]	Modal information (G code: group 1)
<2>	#4201	[#_ACTG[1]]	
<3>	#4401	[#_INTG[1]]	

Category	Variable number	Variable name	Description
<1>	#4002	[#_BUFG[2]]	Modal information (G code: group 2)
<2>	#4202	[#_ACTG[2]]	
<3>	#4402	[#_INTG[2]]	
:	:	:	:
<1>	#4030	[#_BUFG[30]]	Modal information (G code: group 30)
<2>	#4230	[#_ACTG[30]]	
<3>	#4430	[#_INTG[30]]	
<1>	#4102	[#_BUFB]	Modal information (B code)
<2>	#4302	[#_ACTB]	
<3>	#4502	[#_INTB]	
<1>	#4107	[#_BUFD]	Modal information (D code)
<2>	#4307	[#_ACTD]	
<3>	#4507	[#_INTD]	
<1>	#4108	[#_BUFE]	Modal information (E code)
<2>	#4308	[#_ACTE]	
<3>	#4508	[#_INTE]	
<1>	#4109	[#_BUFF]	Modal information (F code)
<2>	#4309	[#_ACTF]	
<3>	#4509	[#_INTF]	
<1>	#4111	[#_BUFH]	Modal information (H code)
<2>	#4311	[#_ACTH]	
<3>	#4511	[#_INTH]	
<1>	#4113	[#_BUFM]	Modal information (M code)
<2>	#4313	[#_ACTM]	
<3>	#4513	[#_INTM]	
<1>	#4114	[#_BUFN]	Modal information (sequence number N)
<2>	#4314	[#_ACTN]	
<3>	#4514	[#_INTN]	
<1>	#4115	[#_BUFO]	Modal information (sequence number O)
<2>	#4315	[#_ACTO]	
<3>	#4515	[#_INTO]	
<1>	#4119	[#_BUFS]	Modal information (S code)
<2>	#4319	[#_ACTS]	
<3>	#4519	[#_INTS]	
<1>	#4120	[#_BUFT]	Modal information (T code)
<2>	#4320	[#_ACTT]	
<3>	#4520	[#_INTT]	
<1>	#4130	[#_BUFWZP]	Modal information (additional workpiece coordinate system number P)
<2>	#4330	[#_ACTWZP]	
<3>	#4530	[#_INTWZP]	

T

(Category: <1> Previous block, <2> Running block, <3> Interrupted block)

Category	Variable number	Variable name	Description
<1>	#4001	[#_BUFG[1]]	Modal information (G code: group 1)
<2>	#4201	[#_ACTG[1]]	
<3>	#4401	[#_INTG[1]]	
<1>	#4002	[#_BUFG[2]]	Modal information (G code: group 2)
<2>	#4202	[#_ACTG[2]]	
<3>	#4402	[#_INTG[2]]	
:	:	:	:
<1>	#4030	[#_BUFG[30]]	Modal information (G code: group 30)
<2>	#4230	[#_ACTG[30]]	
<3>	#4430	[#_INTG[30]]	

Category	Variable number	Variable name	Description
<1>	#4108	[#_BUFE]	Modal information (E code)
<2>	#4308	[#_ACTE]	
<3>	#4508	[#_INTE]	
<1>	#4109	[#_BUFF]	Modal information (F code)
<2>	#4309	[#_ACTF]	
<3>	#4509	[#_INTF]	
<1>	#4113	[#_BUFM]	Modal information (M code)
<2>	#4313	[#_ACTM]	
<3>	#4513	[#_INTM]	
<1>	#4114	[#_BUFN]	Modal information (sequence number N)
<2>	#4314	[#_ACTN]	
<3>	#4514	[#_INTN]	
<1>	#4115	[#_BUFO]	Modal information (sequence number O)
<2>	#4315	[#_ACTO]	
<3>	#4515	[#_INTO]	
<1>	#4119	[#_BUFS]	Modal information (S code)
<2>	#4319	[#_ACTS]	
<3>	#4519	[#_INTS]	
<1>	#4120	[#_BUFT]	Modal information (T code)
<2>	#4320	[#_ACTT]	
<3>	#4520	[#_INTT]	
<1>	#4130	[#_BUFWZP]	Modal information (additional workpiece coordinate system number P)
<2>	#4330	[#_ACTWZP]	
<3>	#4530	[#_INTWZP]	

NOTE

1 Previous block and running block
Since the CNC reads the block that is ahead of the block currently being executed by the machining program, the block being retrieved by the CNC is normally different from that currently being executed. The previous block indicates the block that is ahead of the block being retrieved by the CNC, that is, the block that is ahead of the program block in which #4001 to #4130 are specified.

2 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.

[Example] O1234 ;
N10 G00 X200. Y200. ;
N20 G01 X1000. Y1000. F10. ;
:
:
N50 G00 X500. Y500. ;
N60 #1 = #4001 ;

Assume that the CNC is currently executing N20. If the CNC retrieved and processed the blocks up to N60 as shown above, the running block is N20 and the previous block is N50. Therefore, group 1 modal information in the running block is G01 and group 1 modal information in the previous block is G00.
When N60 #1 = #4201, #1 = 1.
When N60 #1 = #4001, #1 = 0.

- Position information #5001-#5080, #100001-#100200 (Attribute: R)

The end position of the previous block, the specified current position (for the machine coordinate system and workpiece coordinate system), and the skip signal position can be obtained by reading the values of system variables #5001 to #5080 or #100001 to -#100200.

Variable number	Variable name	Position information	Coordinate system	Tool position/tool length/cutter compensation	Reading operation during movement
#5001 #5002 : #5020	[#_ABSIO[1]] [#_ABSIO[2]] : [#_ABSIO[20]]	1st axis block end point position 2nd axis block end point position : 20th axis block end point position	Workpiece coordinate system	Not included	Enabled
#100001 #100002 : #100050	[#_ABSIO[1]] [#_ABSIO[2]] : [#_ABSIO[50]]	1st axis block end point position 2nd axis block end point position : 50th axis block end point position			
#5021 #5022 : #5040	[#_ABSMT[1]] [#_ABSMT[2]] : [#_ABSMT[20]]	1st axis current position 2nd axis current position : 20th axis current position	Machine coordinate system	Included	Disabled
#100051 #100052 : #100100	[#_ABSMT[1]] [#_ABSMT[2]] : [#_ABSMT[50]]	1st axis current position 2nd axis current position : 50th axis current position			
#5041 #5042 : #5060	[#_ABSOT[1]] [#_ABSOT[2]] : [#_ABSOT[20]]	1st axis current position 2nd axis current position : 20th axis current position	Workpiece coordinate system	Included	Disabled
#100101 #100102 : #100150	[#_ABSOT[1]] [#_ABSOT[2]] : [#_ABSOT[50]]	1st axis current position 2nd axis current position : 50th axis current position			
#5061 #5062 : #5080	[#_ABSKP[1]] [#_ABSKP[2]] : [#_ABSKP[20]]	1st axis skip position 2nd axis skip position : 20th axis skip position	Workpiece coordinate system	Included	Enabled
#100151 #100152 : #100200	[#_ABSKP[1]] [#_ABSKP[2]] : [#_ABSKP[50]]	1st axis skip position 2nd axis skip position : 50th axis skip position			

NOTE

1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The position information for 20th or earlier axis can be used with #5001 to #5080.
3 The block end point position (ABSIO) of the skip (G31) is the position where the skip signal is turned on. If the skip signal is not turned on, the position is the end position of the block.
4 "Read operation during movement is disabled" means that the accurate reading of values during movement is not guaranteed.

- Tool length compensation value #5081-#5100, #100201-#100250 (Attribute: R)

M

Tool length compensation in the block currently being executed can be obtained for each axis by reading system variables #5081 to #5100 or #100201 to #100250.

Variable number	Variable name	Position information	Read operation during movement
#5081	[#_TOFS[1]]	1st axis tool length compensation value	Disabled
#5082	[#_TOFS[2]]	2nd axis tool length compensation value	
:	:	:	
#5100	[#_TOFS[20]]	20th axis tool length compensation value	
#100201	[#_TOFS[1]]	1st axis tool length compensation value	
#100202	[#_TOFS[2]]	2nd axis tool length compensation value	
:	:	:	
#100250	[#_TOFS[50]]	50th axis tool length compensation value	

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The tool length compensation for 20th or earlier axis can also be used with #5081 to #5100.

- Tool offset #5081-#5083, #5121-#5123 (Attribute: R)
#100201-#100250, #100901-#100950

T

Tool offset in the block currently being executed can be obtained for each axis by reading system variables #5081 to #5083 or #5121 to #5123. (X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes)

<1> Without tool geometry/wear compensation memory

Variable number	Variable name	Position information	Read operation during movement
#5081	[#_TOFSWX]	X-axis tool offset value	Disabled
#5082	[#_TOFSWZ]	Z-axis tool offset value	
#5083	[#_TOFSWY]	Y-axis tool offset value	
#5084	[#_TOFS[4]]	4th axis tool offset value	
:	:	:	
#5100	[#_TOFS[20]]	20th axis tool offset value	
#100201	[#_TOFSWX]	X-axis tool offset value	
#100202	[#_TOFSWZ]	Z-axis tool offset value	
#100203	[#_TOFSWY]	Y-axis tool offset value	
#100204	[#_TOFS[4]]	4th axis tool offset value	
:	:	:	
#100250	[#_TOFS[50]]	50th axis tool offset value	

<2> With tool geometry/wear compensation memory

Variable number	Variable name	Position information	Read operation during movement
#5081	[#_TOFSWX]	X-axis tool offset value (geometry)	Disabled
#5082	[#_TOFSWZ]	Z-axis tool offset value (geometry)	
#5083	[#_TOFSWY]	Y-axis tool offset value (geometry)	
#5084	[#_TOFS[4]]	4th axis tool offset value (geometry)	
:	:	:	
#5100	[#_TOFS[20]]	20th axis tool offset value (geometry)	
#100201	[#_TOFSWX]	X-axis tool offset value (geometry)	
#100202	[#_TOFSWZ]	Z-axis tool offset value (geometry)	
#100203	[#_TOFSWY]	Y-axis tool offset value (geometry)	
#100204	[#_TOFS[4]]	4th axis tool offset value (geometry)	
:	:	:	
#100250	[#_TOFS[50]]	50th axis tool offset value (geometry)	
#5121	[#_TOFSGX]	X-axis tool offset value (geometry)	Disabled
#5122	[#_TOFSGZ]	Z-axis tool offset value (geometry)	
#5123	[#_TOFSGY]	Y-axis tool offset value (geometry)	
#5124	[#_TOFSG[4]]	4th axis tool offset value (geometry)	
:	:	:	
#5140	[#_TOFSG[20]]	20th axis tool offset value (geometry)	
#100901	[#_TOFSGX]	X-axis tool offset value (geometry)	
#100902	[#_TOFSGZ]	Z-axis tool offset value (geometry)	
#100903	[#_TOFSGY]	Y-axis tool offset value (geometry)	
#100904	[#_TOFSG[4]]	4th axis tool offset value (geometry)	
:	:	:	
#100950	[#_TOFSG[50]]	50th axis tool offset value (geometry)	

When tool geometry/wear compensation memory exists, the system variable values vary as follows depending on bit 2 (LWT) of parameter No. 5002 and bit 4 (LGT) of parameter No. 5002.

Variable number	LWT=0 LGT=0	LWT=1 LGT=0	LWT=0 LGT=1	LWT=1 LGT=1
#5081 #5082 #5083	Wear compensation	0	Wear compensation	Wear compensation
#5121 #5122 #5123	Geometry compensation	Wear compensation + Geometry compensation	Geometry compensation	Geometry compensation

NOTE
1 The set value is read as the tool offset regardless of bit 1 (ORC) of parameter No.5004 and bit 0 (OWD) of parameter No.5040.
2 To read the tool offset (geometry) using #5121 to #5123, set bit 2 (VHD) of parameter No.6004 to 0.

- Servo position deviation #5101-#5120, #100251-#100300 (Attribute: R)

The servo position deviation for each axis can be obtained by reading system variables #5101 to #5120 or #100251 to #100300.

Variable number	Variable name	Position information	Read operation during movement
#5101	[#_SVERR[1]]	1st axis servo position deviation	Disabled
#5102	[#_SVERR[2]]	2nd axis servo position deviation	
:	:	:	
#5120	[#_SVERR[20]]	20th axis servo position deviation	
#100251	[#_SVERR[1]]	1st axis servo position deviation	
#100252	[#_SVERR[2]]	2nd axis servo position deviation	
:	:	:	
#100300	[#_SVERR[50]]	50th axis servo position deviation	

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The servo position deviation for 20th or earlier axis can be used with #5101 to #5120.

- Manual handle interruption #5121-#5140, #100651-#100700 (Attribute: R)

The manual handle interruption for each axis can be obtained by reading system variables #5121 to #5140 or #100651 to #100700.

Variable number	Variable name	Position information	Read operation during movement
#5121	[#_MIRTP[1]]	1st axis manual handle interruption	Disabled
#5122	[#_MIRTP[2]]	2nd axis manual handle interruption	
:	:	:	
#5140	[#_MIRTP[20]]	20th axis manual handle interruption	
#100651	[#_MIRTP[1]]	1st axis manual handle interruption	
#100652	[#_MIRTP[2]]	2nd axis manual handle interruption	
:	:	:	
#100700	[#_MIRTP[50]]	50th axis manual handle interruption	

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The manual handle interruption for 20th or earlier axis can be used with #5121 to #5140.

T

NOTE
#5121 to #5140 are enabled only when bit 2 (VHD) of parameter No.6004 is set to 1.

- Distance to go #5181-#5200, #100801-#100850 (Attribute: R)

The distance to go value for each axis can be obtained by reading system variables #5181 to #5200 or #100801 to #100850.

Variable number	Variable name	Position information	Read operation during movement
#5181	[#_DIST[1]]	1st axis distance to go value	Disabled
#5182	[#_DIST[2]]	2nd axis distance to go value	
:	:	:	
#5200	[#_DIST[20]]	20th axis distance to go value	
#100801	[#_DIST[1]]	1st axis distance to go value	
#100802	[#_DIST[2]]	2nd axis distance to go value	
:	:	:	
#100850	[#_DIST[50]]	50th axis distance to go value	

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The distance to go values for 20th or earlier axis can be used with #5181 to #5200.

- Workpiece coordinate system shift value #2501, #2601 (Attribute: R/W)

T

The X-axis workpiece coordinate system shift value can be obtained by system variable #2501 while Z-axis workpiece coordinate system shift value by system variable #2601. Each workpiece coordinate system shift value can be obtained by system variable #100751-#100800. The X-axis or Z-axis workpiece coordinate system shift value can be changed by assigning values to the system variables. (X-axis: X-axis of basic three axes, Z-axis: Z-axis of basic three axes, Y-axis: Y-axis of basic three axes)

Variable number	Variable name	Description
#2501	[#_WZ_SFTX]	X-axis workpiece shift value
#2601	[#_WZ_SFTZ]	Z-axis workpiece shift value
#100751	[#_WZ_SFT[1]]	1st axis workpiece shift value
#100752	[#_WZ_SFT [2]]	2nd axis workpiece shift value
:	:	:
#100800	[#_WZ_SFT [50]]	50th axis workpiece shift value

NOTE
1 The value of a variable with a number greater than the number of controlled axes is undefined.
2 The X-axis of three basic axes can also be used in system variable #2501, and the Z-axis of three basic axes can also be used in system variable #2601.

- Workpiece origin offset value #5201-#5340, #100301-#100650 (Attribute: R/W)

The workpiece origin offset value can be obtained by reading system variables #5201 to #5340 or #100301 to #100650. The offset value can also be changed by assigning values to the system variables.

Variable number	Variable name	Controlled axis	Workpiece coordinate system
#5201	[#_WZCMN[1]]	1st axis external workpiece origin offset value	External workpiece origin offset value (applied to all coordinate systems)
#5202	[#_WZCMN[2]]	2nd axis external workpiece origin offset value	
:	:	:	
#5220	[#_WZCMN[20]]	20th axis external workpiece origin offset value	

Variable number	Variable name	Controlled axis	Workpiece coordinate system
#5221	[#_WZG54[1]]	1st axis workpiece origin offset value	G54
#5222	[#_WZG54[2]]	2nd axis workpiece origin offset value	
:	:	:	
#5240	[#_WZG54[20]]	20th axis workpiece origin offset value	
#5241	[#_WZG55[1]]	1st axis workpiece origin offset value	G55
#5242	[#_WZG55[2]]	2nd axis workpiece origin offset value	
:	:	:	
#5260	[#_WZG55[20]]	20th axis workpiece origin offset value	
#5261	[#_WZG56[1]]	1st axis workpiece origin offset value	G56
#5262	[#_WZG56[2]]	2nd axis workpiece origin offset value	
:	:	:	
#5280	[#_WZG56[20]]	20th axis workpiece origin offset value	
#5281	[#_WZG57[1]]	1st axis workpiece origin offset value	G57
#5282	[#_WZG57[2]]	2nd axis workpiece origin offset value	
:	:	:	
#5300	[#_WZG57[20]]	20th axis workpiece origin offset value	
#5301	[#_WZG58[1]]	1st axis workpiece origin offset value	G58
#5302	[#_WZG58[2]]	2nd axis workpiece origin offset value	
:	:	:	
#5320	[#_WZG58[20]]	20th axis workpiece origin offset value	
#5321	[#_WZG59[1]]	1st axis workpiece origin offset value	G59
#5322	[#_WZG59[2]]	2nd axis workpiece origin offset value	
:	:	:	
#5340	[#_WZG59[20]]	20th axis workpiece origin offset value	

Variable number	Variable name	Controlled axis	Workpiece coordinate system
#100301	[#_WZCMN[1]]	1st axis external workpiece origin offset value	External workpiece origin offset value (applied to all coordinate systems)
#100302	[#_WZCMN[2]]	2nd axis external workpiece origin offset value	
:	:	:	
#100350	[#_WZCMN[50]]	50th axis external workpiece origin offset value	
#100351	[#_WZG54[1]]	1st axis workpiece origin offset value	G54
#100352	[#_WZG54[2]]	2nd axis workpiece origin offset value	
:	:	:	
#100400	[#_WZG54[50]]	50th axis workpiece origin offset value	
#100401	[#_WZG55[1]]	1st axis workpiece origin offset value	G55
#100402	[#_WZG55[2]]	2nd axis workpiece origin offset value	
:	:	:	
#100450	[#_WZG55[50]]	50th axis workpiece origin offset value	
#100451	[#_WZG56[1]]	1st axis workpiece origin offset	G56
#100452	[#_WZG56[2]]	2nd axis workpiece origin offset	
:	:	:	
#100500	[#_WZG56[50]]	50th axis workpiece origin offset	
#100501	[#_WZG57[1]]	1st axis workpiece origin offset	G57
#100502	[#_WZG57[2]]	2nd axis workpiece origin offset	
:	:	:	
#100550	[#_WZG57[50]]	50th axis workpiece origin offset	
#100551	[#_WZG58[1]]	1st axis workpiece origin offset	G58
#100552	[#_WZG58[2]]	2nd axis workpiece origin offset	
:	:	:	
#100600	[#_WZG58[50]]	50th axis workpiece origin offset	
#100601	[#_WZG59[1]]	1st axis workpiece origin offset	G59
#100602	[#_WZG59[2]]	2nd axis workpiece origin offset	
:	:	:	
#100650	[#_WZG59[50]]	50th axis workpiece origin offset	

M

The following variables can also be used when bit 5 (D15) of parameter No. 6004 is set to 0:

Axis	Function	Variable number
1st axis	External workpiece origin offset value	#2500
	G54 workpiece origin offset value	#2501
	G55 workpiece origin offset value	#2502
	G56 workpiece origin offset value	#2503
	G57 workpiece origin offset value	#2504
	G58 workpiece origin offset value	#2505
	G59 workpiece origin offset value	#2506
2nd axis	External workpiece origin offset value	#2600
	G54 workpiece origin offset value	#2601
	G55 workpiece origin offset value	#2602
	G56 workpiece origin offset value	#2603
	G57 workpiece origin offset value	#2604
	G58 workpiece origin offset value	#2605
	G59 workpiece origin offset value	#2606
3rd axis	External workpiece origin offset value	#2700
	G54 workpiece origin offset value	#2701
	G55 workpiece origin offset value	#2702
	G56 workpiece origin offset value	#2703
	G57 workpiece origin offset value	#2704
	G58 workpiece origin offset value	#2705
	G59 workpiece origin offset value	#2706
4th axis	External workpiece origin offset value	#2800
	G54 workpiece origin offset value	#2801
	G55 workpiece origin offset value	#2802
	G56 workpiece origin offset value	#2803
	G57 workpiece origin offset value	#2804
	G58 workpiece origin offset value	#2805
	G59 workpiece origin offset value	#2806

T

The following variables can be used to maintain compatibility with conventional models.

Axis	Function	Variable number
1st axis	External workpiece origin offset value	#2550
	G54 workpiece origin offset value	#2551
	G55 workpiece origin offset value	#2552
	G56 workpiece origin offset value	#2553
	G57 workpiece origin offset value	#2554
	G58 workpiece origin offset value	#2555
	G59 workpiece origin offset value	#2556
2nd axis	External workpiece origin offset value	#2650
	G54 workpiece origin offset value	#2651
	G55 workpiece origin offset value	#2652
	G56 workpiece origin offset value	#2653
	G57 workpiece origin offset value	#2654
	G58 workpiece origin offset value	#2655
	G59 workpiece origin offset value	#2656

Axis	Function	Variable number
3rd axis	External workpiece origin offset value	#2750
	G54 workpiece origin offset value	#2751
	G55 workpiece origin offset value	#2752
	G56 workpiece origin offset value	#2753
	G57 workpiece origin offset value	#2754
	G58 workpiece origin offset value	#2755
	G59 workpiece origin offset value	#2756
4th axis	External workpiece origin offset value	#2850
	G54 workpiece origin offset value	#2851
	G55 workpiece origin offset value	#2852
	G56 workpiece origin offset value	#2853
	G57 workpiece origin offset value	#2854
	G58 workpiece origin offset value	#2855
	G59 workpiece origin offset value	#2856

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The workpiece origin offset value for 20th or earlier axis can be used with #5201 to #5340.

M

NOTE
To use variables #2500 to #2806, #5201 to #5340, and #100301 to #100650, optional variables for the workpiece coordinate systems are necessary.

T

NOTE
To use variables #2550 to #2856, #5201 to #5340, and #100301 to #100650, optional variables for the workpiece coordinate systems are necessary.

- Workpiece origin offset value of the additional workpiece coordinate system

#7001-#7960, #101001-#116000 (Attribute: R/W)

M

#14001-#20000 (Attribute: R/W)

The workpiece origin offset value of the additional workpiece coordinate system can be obtained by reading system variables #7001 to #7960, #14001 to #20000, #101001 to #116000. The offset value can also be changed by assigning values to the system variables.

Variable number	Variable name	Controlled axis	Additional workpiece system number
#7001	[#_WZP1[1]]	1st axis workpiece origin offset value	
#7002	[#_WZP1[2]]	2nd axis workpiece origin offset value	1
:	:	:	(G54.1 P1)
#7020	[#_WZP1[20]]	20th axis workpiece origin offset value	
#7021	[#_WZP2[1]]	1st axis workpiece origin offset value	
#7022	[#_WZP2[2]]	2nd axis workpiece origin offset value	2
:	:	:	(G54.1 P2)
#7040	[#_WZP2[20]]	20th axis workpiece origin offset value	

Variable number	Variable name	Controlled axis	Additional workpiece system number
#7041	[#_WZP3[1]]	1st axis workpiece origin offset value	
#7042	[#_WZP3[2]]	2nd axis workpiece origin offset value	3
:	:	:	(G54.1 P3)
#7060	[#_WZP3[20]]	20th axis workpiece origin offset value	
:	:	:	:
#7941	[#_WZP48[1]]	1st axis workpiece origin offset value	
#7942	[#_WZP48[2]]	2nd axis workpiece origin offset value	48
:	:	:	(G54.1 P48)
#7960	[#_WZP48[20]]	20th axis workpiece origin offset value	

System variable number = 7000 + (Coordinate system number -1) × 20 + Axis number
Coordinate number: 1 to 48
Axis number: 1 to 20

M

Variable number	Variable name	Controlled axis	Additional workpiece system number
#14001	[#_WZP1[1]]	1st axis workpiece origin offset value	
#14002	[#_WZP1[2]]	2nd axis workpiece origin offset value	1
:	:	:	(G54.1 P1)
#14020	[#_WZP1[20]]	20th axis workpiece origin offset value	
#14021	[#_WZP2[1]]	1st axis workpiece origin offset value	
#14022	[#_WZP2[2]]	2nd axis workpiece origin offset value	2
:	:	:	(G54.1 P2)
#14040	[#_WZP2[20]]	20th axis workpiece origin offset value	
#14041	[#_WZP3[1]]	1st axis workpiece origin offset value	
#14042	[#_WZP3[2]]	2nd axis workpiece origin offset value	3
:	:	:	(G54.1 P3)
#14060	[#_WZP3[20]]	20th axis workpiece origin offset value	
:	:	:	:
#19981	[#_WZP300[1]]	1st axis workpiece origin offset value	
#19982	[#_WZP300[2]]	2nd axis workpiece origin offset value	300
:	:	:	(G54.1 P300)
#20000	[#_WZP300[20]]	20th axis workpiece origin offset value	

System variable number = 14000 + (Coordinate system number -1) × 20 + Axis number
Coordinate number: 1 to 300
Axis number: 1 to 20

Variable number	Variable name	Controlled axis	Additional workpiece system number
#101001	[#_WZP1[1]]	1st axis workpiece origin offset value	
#101002	[#_WZP1[2]]	2nd axis workpiece origin offset value	1
:	:	:	(G54.1 P1)
#101050	[#_WZP1[50]]	50th axis workpiece origin offset value	
#101051	[#_WZP2[1]]	1st axis workpiece origin offset value	
#101052	[#_WZP2[2]]	2nd axis workpiece origin offset value	2
:	:	:	(G54.1 P2)
#101100	[#_WZP2[50]]	50th axis workpiece origin offset value	
#101101	[#_WZP3[1]]	1st axis workpiece origin offset value	
#101102	[#_WZP3[2]]	2nd axis workpiece origin offset value	3
:	:	:	(G54.1 P3)
#101150	[#_WZP3[50]]	50th axis workpiece origin offset value	

Variable number	Variable name	Controlled axis	Additional workpiece system number
:	:	:	:
#115951	[#_WZP300[1]]	1st axis workpiece origin offset value	
#115952	[#_WZP300[2]]	2nd axis workpiece origin offset value	300
:	:	:	(G54.1 P300)
#116000	[#_WZP300[50]]	50th axis workpiece origin offset value	

System variable number = 101000 + (Coordinate system number -1) × 50 + Axis number
Coordinate number: 1 to 300
Axis number: 1 to 50

M

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The workpiece origin offset of additional workpiece coordinate system for 20th or earlier axis can be used with #7001 to #7960 or #14001 to #20000.
3 Optional variables for 48 additional workpiece coordinate systems are #7001 to #7960 (G54.1 P1 to G54.1 P48). Optional variables for 300 additional workpiece coordinate systems are #14001 to #20000, and #101001 to #116000 (G54.1 P1 to G54.1 P300). With these variables, #7001 to #7960 can also be used.

T

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 Optional variables for 48 additional workpiece coordinate systems are #7001 to #7960 (G54.1 P1 to G54.1 P48).
3 Optional variables for 300 additional workpiece coordinate systems are #101001 to #116000 (G54.1 P1 to G54.1 P300). With these variables, #7001 to #7960 can also be used.

- Skip position (detection unit) #5421-#5440, #100701-#100750 (Attribute: R)

The skip position with the detection unit can be obtained by reading system variables #5421 to #5440 or #100701 to #100750. The values of these variables are integers, and values in the workpiece coordinate system.

Variable number	Variable name	Position information	Read operation during movement
#5421	[#_SKPDTC[1]]	1st axis skip position (detection unit)	Disabled
#5422	[#_SKPDTC[2]]	2nd axis skip position (detection unit)	
:	:	:	
#5440	[#_SKPDTC[20]]	20th axis skip position (detection unit)	
#100151	[#_SKPDTC[1]]	1st axis skip position (detection unit)	
#100152	[#_SKPDTC[2]]	2nd axis skip position (detection unit)	
:	:	:	
#100200	[#_SKPDTC[50]]	50th axis skip position (detection unit)	

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The skip position (detection unit) for 20th or earlier axis can be used with #5421 to #5440.
3 To specify these variables, set bit 2 (DSK) of parameter No. 6210 to 1. (If an attempt is made to specify them when DSK is 0, alarm PS0115 is issued.)

- Reference fixture offset number being selected #5500 (Attribute: R)

M

The reference fixture offset number being selected can be read by reading system variables #5500.

Variable number	Variable name	Description
#5500	[#_FOFSP]	Reference fixture offset number being selected

- Reference fixture offset value being selected #5501-#5520, #117001-#117050 (Attribute: R)

M

The reference fixture offset being selected can be read by reading system variables #5501 to #5520 or #117001# to 117050.

Variable number	Variable name	Description
#5501	[#_FOFSVAL[1]]	1st axis reference fixture offset value being selected
#5502	[#_FOFSVAL[2]]	2nd axis reference fixture offset value being selected
:	:	:
#5520	[#_FOFSVAL[20]]	20th axis reference fixture offset value being selected
#117001	[#_FOFSVAL[1]]	1st axis reference fixture offset value being selected
#117002	[#_FOFSVAL[2]]	2nd axis reference fixture offset value being selected
:	:	:
#117050	[#_FOFSVAL[50]]	50th axis reference fixture offset value being selected

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The reference fixture offset for 20th or earlier axis can be used with #5501 to #5520.

- Reference fixture offset value #5521-#5680, #117051-#117450 (Attribute: R/W)

M

The reference fixture offset values in the rotary table dynamic fixture offset function can be obtained by reading system variables #5521 to #5680 or #117051 to #117450. The reference fixture offset values can also be changed by assigning values to the system values.

Variable number	Variable name	Controlled axis	Fixture offset number
#5521	[#_FOFS1[1]]	1st axis reference fixture offset value being selected	
#5522	[#_FOFS1[2]]	2nd axis reference fixture offset value being selected	1
:	:	:	(G54.2 P1)
#5540	[#_FOFS1[20]]	20th axis reference fixture offset value being selected	
#5541	[#_FOFS2[1]]	1st axis reference fixture offset value being selected	
#5542	[#_FOFS2[2]]	2nd axis reference fixture offset value being selected	2
:	:	:	(G54.2 P2)
#5560	[#_FOFS2[20]]	20th axis reference fixture offset value being selected	
:	:	:	:

Variable number	Variable name	Controlled axis	Fixture offset number
#5661	[#_FOFS8[1]]	1st axis reference fixture offset value being selected	
#5662	[#_FOFS8[2]]	2nd axis reference fixture offset value being selected	8
:	:	:	(G54.2 P8)
#5680	[#_FOFS8[20]]	20th axis reference fixture offset value being selected	
#117051	[#_FOFS1[1]]	1st axis reference fixture offset value being selected	
#117052	[#_FOFS1[2]]	2nd axis reference fixture offset value being selected	1
:	:	:	(G54.2 P1)
#117100	[#_FOFS1[50]]	50th axis reference fixture offset value being selected	
#117101	[#_FOFS2[1]]	1st axis reference fixture offset value being selected	
#117102	[#_FOFS2[2]]	2nd axis reference fixture offset value being selected	2
:	:	:	(G54.2 P2)
#117150	[#_FOFS2[50]]	50th axis reference fixture offset value being selected	
:	:	:	:
#117401	[#_FOFS8[1]]	1st axis reference fixture offset value being selected	
#117402	[#_FOFS8[2]]	2nd axis reference fixture offset value being selected	8
:	:	:	(G54.2 P8)
#117450	[#_FOFS8[50]]	50th axis reference fixture offset value being selected	

NOTE
1 When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.
2 The reference fixture offset values for 20th or earlier axis can be used with #5521 to #5580.

- Second geometry tool offset value #5801-#5896, #27001-#29999 (Attribute: R/W)

T

By reading the values of system variables #5801 to #5896 and #27001 to #29999, it is possible to determine the second geometry tool offset value, and by assigning values to the system variables, it is possible to change the second geometry tool offset value.

- For compensation numbers up to 32

Compensation number	Variable number	Variable name	Attribute	Description
1	#5801	[#_OFSX2G[1]]	R/W	Second geometry tool offset X-axis compensation value
:	:	:	:	
32	#5832	[#_OFSX2G[32]]	R/W	
1	#5833	[#_OFSZ2G[1]]	R/W	Second geometry tool offset Z-axis compensation value
:	:	:	:	
32	#5864	[#_OFSZ2G[32]]	R/W	
1	#5865	[#_OFSY2G[1]]	R/W	Second geometry tool offset Y-axis compensation value
:	:	:	:	
32	#5896	[#_OFSY2G[32]]	R/W	

- For compensation numbers greater than 32 (Compensation values with compensation numbers of up to 32 can also be used in variables #5801 to #5896.)

Compensation number	Variable number	Variable name	Attribute	Description
1	#27001	[#_OFSX2G[1]]	R/W	Second geometry tool offset X-axis compensation value
:	:	:	:	
999	#27999	[#_OFSX2G[999]]	R/W	

Compensation number	Variable number	Variable name	Attribute	Description
1	#28001	[#_OFSZ2G[1]]	R/W	Second geometry tool offset Z-axis compensation value
:	:	:	:	
999	#28999	[#_OFSZ2G[999]]	R/W	
1	#29001	[#_OFSY2G[1]]	R/W	Second geometry tool offset Y-axis compensation value
:	:	:	:	
999	#29999	[#_OFSY2G[999]]	R/W	

- Feedrate reduction ratio for rapid traverse overlap #100851-#100900 (Attribute: R/W)

The feedrate reduction ratio for rapid traverse overlap can also be changed by setting values to the system variables #100851 to #100900.

Variable number	Variable name	Controlled axis
#100851	[#_ROVLP[1]]	1st axis feedrate reduction ratio for rapid traverse overlap
#100852	[#_ROVLP[2]]	2nd axis feedrate reduction ratio for rapid traverse overlap
:	:	:
#100900	[#_ROVLP[50]]	50th axis feedrate reduction ratio for rapid traverse overlap

- States of serial spindles #100951 to #100954 (Attribute: R)

Using system variables #100951 to #100954, it is possible to determine the states of serial spindles. The relationships between the values of the variables and the states of spindles are as follows:

1 : Normal operation
2 : Orientation
3 : Synchronous control
4 : Rigid tapping
5 : Cs contour control

Variable number	Variable name	Description
#100951	[#_SPSTAT[1]]	State of the first spindle on the path
#100952	[#_SPSTAT[2]]	State of the second spindle on the path
#100953	[#_SPSTAT[3]]	State of the third spindle on the path
#100954	[#_SPSTAT[4]]	State of the fourth spindle on the path

- Dynamic reference tool compensation value #118051-#118450 (Attribute: R/W)

M

The dynamic reference tool compensation value in the rotary head dynamic tool compensation function can be obtained by reading system variables #118051 to #118450. The dynamic reference tool compensation value can also be obtained by assigning values to the system variables.

Variable number	Variable name	Controlled axis	Dynamic tool offset number
#118051	[#_DOFS1[1]]	1st axis dynamic reference tool compensation value	1 (G43.2H1)
#118052	[#_DOFS1[2]]	2nd axis dynamic reference tool compensation value	
:	:	:	
#118100	[#_DOFS1[50]]	50th axis dynamic reference tool compensation value	
#118101	[#_DOFS2[1]]	1st axis dynamic reference tool compensation value	2 (G43.2H2)
#118102	[#_DOFS2[2]]	2nd axis dynamic reference tool compensation value	
:	:	:	
#118150	[#_DOFS2[50]]	50th axis dynamic reference tool compensation value	
#118151	[#_DOFS3[1]]	1st axis dynamic reference tool compensation value	3 (G43.2H3)
#118152	[#_DOFS3[2]]	2nd axis dynamic reference tool compensation value	
:	:	:	
#118200	[#_DOFS3[50]]	50th axis dynamic reference tool compensation value	

Variable number	Variable name	Controlled axis	Dynamic tool offset number
#118201	[#_DOFS4[1]]	1st axis dynamic reference tool compensation value	
#118202	[#_DOFS4[2]]	2nd axis dynamic reference tool compensation value	4
:	:	:	(G43.2H4)
#118250	[#_DOFS4[50]]	50th axis dynamic reference tool compensation value	
#118251	[#_DOFS5[1]]	1st axis dynamic reference tool compensation value	
#118252	[#_DOFS5[2]]	2nd axis dynamic reference tool compensation value	5
:	:	:	(G43.2H5)
#118300	[#_DOFS5[50]]	50th axis dynamic reference tool compensation value	
:	:	:	:
#118401	[#_DOFS8[1]]	1st axis dynamic reference tool compensation value	
#118402	[#_DOFS8[2]]	2nd axis dynamic reference tool compensation value	8
:	:	:	(G43.2H8)
#118450	[#_DOFS8[50]]	50th axis dynamic reference tool compensation value	

NOTE
When variables exceeding the number of control axes are specified, the alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.

- Local coordinate system offset value #118501-#118550 (Attribute: R)

Using system variables #118501 to #118550, it is possible to determine the local coordinate system (G52) offset value.

Variable number	Variable name	Description
#118501	[#_LCLOFS[1]]	1st axis local coordinate system offset value
#118502	[#_LCLOFS[2]]	2nd axis local coordinate system offset value
:	:	:
#118550	[#_LCLOFS[50]]	50th axis local coordinate system offset value

- Switching between P-CODE variables and system variables (#10000 or later) #8570 (Attribute: R/W)

This system variable allows read/write operations of P-CODE variables (#10000 to #89999) for the macro executor function. For details on P-CODE variables, refer to the Macro Executor Programming Manual (B-63943EN-2).

System variable #8570 can be used to make variables #10000 or later correspond to either P-CODE variables or system variables.

#8570 setting	Specified variable	Corresponding variable
#8570 = 1	#10000	P-CODE variables (#10000)
	:	:
	#89999	P-CODE variables (#89999)
#8570 = 0	#10000	System variables (#10000)
	:	:
	#89999	System variables (#89999)

Example

```
#8570 = 0 ;
#10001 = 123 ; →  Writing to system variable #10001 (tool compensation)
#8570 = 1 ;
#10001 = 456 ; →  Writing to P-CODE variable #10001 (tool compensation)
```

NOTE
1 Variable #8570 can be used only when the macro executor function is enabled.
2 System variables (#10000 or later) always correspond to system variables specified by their variable names even when #8570 is 1.
3 When an attempt is made to access a variable that cannot be used with P-CODE variables (#10000 or later), an alarm PS0115, “VARIABLE NO. OUT OF RANGE” occurs.

16.3 READING AND WRITING VARIABLES FOR ANOTHER PATH

By adding a path number to the high-order 8th and 9th digits of a variable, it is possible to read and write a common variable or a system variable for another path.
For a list of variables that can be read and written, see "List of variables that can be read and written".

Format

#ppxxxxxxx

pp: Path number
Omitted = Local path
1 = First path
: :
10 = Tenth path
xxxxxxx: Variable number

Common variable (#100 to #499(Note), #500 to #999(Note)) or system variable number (1000 and above, 10000 and above, 100000 and above)

NOTE
Available common variables and system variables differ depending on the system configuration.

Example

Machine coordinates for path 1		Machine coordinates for path 2	
X1	123.456	X2	-123.456
Y1	45.670	Z2	78.900
Z1	345.789	C2	45.000

In the case above, if #100=#5023 is executed on path 1, the machine coordinate 345.789 on the third axis on path 1 is read into variable #100.
If #100=#20005023 is executed, the machine coordinate 45.0 on the third axis on path 2 is read into variable #100.

NOTE
1 Variable names cannot be specified.
2 If a path number out of the range is specified, alarm PS0115, “VARIABLE NO. OUT OF RANGE” is issued.
3 Local variables for another path cannot be read or written.
4 To read and write data for the 10th path, set bit 0 (F16) of parameter No. 6008 to 0.
If bit 0 (F16) of parameter No. 6008 is 1, the calculation precision is up to 8th digit. Thus, if data for the 10th path is to be read and written, calculation may not be performed correctly if a calculation command such as #100=#[100005000+#1] is executed.
5 If a real time custom macro is executed, alarm PS0390, “ILLEGAL MACRO VAR” is issued.

Caution

> **⚠ CAUTION**
> System variables include those that affect automatic operation (for example, variables #3000 to #3999), and they affect the operation on another path. Use great caution when writing them.

List of variables that can be read and written

This function can read and write the following variables for another path.
For details of system variables, see the description of variables in the previous section.
R, W, and R/W are variable attributes, and respectively represent READ only, WRITE only, and READ/WRITE.

- Common variable

System variable number	Attribute	Description
#100 to #149(#199) #200 to #499 #500 to #549(#999)	R/W	Available common variables differ depending on the system configuration.

- Interface signals

System variable number	Attribute	Description
#1000 to #1035	R	Interface input signals
#1100 to #1135	R/W	Interface output signals

- Tool compensation value

System variable number	Attribute	Description
#2001 to #2964 #10001 to #19999	R/W	Tool compensation value

- Workpiece coordinate system shift amount

T

System variable number	Attribute	Description
#100751 to #100800	R/W	Workpiece coordinate system shift amount

- Automatic operation or the like

System variable number	Attribute	Description
#3001, #3002	R/W	Clock
#3003	R/W	Control for waiting for a single block stop and the auxiliary function completion signal
#3004	R/W	Feed hold, feedrate override, exact stop check
#3005	R/W	Setting data.
#3007	R	Status of a mirror image (DI and setting)
#3008	R	State during a program restart

- Time

System variable number	Attribute	Description
#3011 to #3012	R	Year/Month/Date, Hour/Minute/Second

- Number of parts

System variable number	Attribute	Description
#3901 to #3902	R/W	Total number of parts, Number of required parts

- Tool compensation memory

M

System variable number	Attribute	Description
#3980	R	Tool compensation memory information

- Main program number

System variable number	Attribute	Description
#4000	R	Main program number

- Modal information

System variable number	Attribute	Description
#4001 to #4530	R	Modal information

- Position information

System variable number	Attribute	Description
#5001 to #5020 #100001 to #100050	R	End point position of the block (workpiece coordinate system)
#5021 to #5040 #100051 to #100100	R	Current position (machine coordinate system)
#5041 to #5060 #100101 to #100150	R	Specified current position (workpiece coordinate system)
#5061 to #5080 #100151 to #100200	R	Skip position (workpiece coordinate system)

- Tool length compensation value

M

System variable number	Attribute	Description
#5081 to #5100 #100201 to #100250	R	Tool length compensation value

- Tool offset value

T

System variable number	Attribute	Description
#5081 to #5083	R	Tool offset value (wear)
#5121 to #5123	R	Tool offset value (geometry)

- Servo position deviation

System variable number	Attribute	Description
#5101 to #5120 #100251 to #100300	R	Servo positional deviation

- Manual handle interruption

System variable number	Attribute	Description
#5121 to #5140 #100651 to #100700	R	Manual handle interruption

- Distance to go

System variable number	Attribute	Description
#5181 to #5200 #100801 to #100850	R	Distance to go

- Workpiece origin offset value, extended workpiece origin offset value

M

System variable number	Attribute	Description
#5201 to #5340 #100301 to #100650	R/W	Workpiece origin offset value
Extended workpiece origin offset value		
#7001 to #7960 #14001 to #20000 #101001 to #116000	R/W	Workpiece origin offset value

T

System variable number	Attribute	Description
#5201 to #5340 #100301 to #100650	R/W	Workpiece origin offset value
Extended workpiece origin offset value		
#7001 to #7960 #101001 to #116000	R/W	Workpiece origin offset value

- Skip position (detection unit)

System variable number	Attribute	Description
#5421 to #5440 #100701 to #100750	R	Skip position (detection unit)

- Second geometry tool offset value

T

System variable number	Attribute	Description
#27001 to #29999	R/W	Second geometry tool offset value

- Others (#8570)

System variable number	Attribute	Description
#8570	R/W	Switching of P-CODE variables/system variables (#10000 and above)

16.4 ARITHMETIC AND LOGIC OPERATION

Various operations can be performed on variables. Program an arithmetic and logic operation in the same way as for a general arithmetic expression.

#i=<expression>

<Expression>

The expression to the right of the arithmetic and logic operation contains constants and/or variables combined by a function or operator. Variables #j and #k below can be replaced with a constant. If a constant used in an expression has no decimal point, it is assumed to end with a decimal point.

Table 16.4 (a) Arithmetic and logic operation

Type of operation	Operation	Description
<1> Definition or replacement	#i=#j	Definition or replacement of a variable
<2> Addition-type operations	#i=#j+#k	Addition
	#i=#j-#k	Subtraction
	#i=#j OR #k	Logical OR (bit by bit of 32 bits)
	#i=#j XOR #k	Exclusive OR (bit by bit of 32 bits)
<3> Multiplication-type operations	#i=#j*#k	Multiplication
	#i=#j/#k	Division
	#i=#j AND #k	Logical AND (bit by bit of 32 bits)
	#i=#j MOD #k	Remainder (A remainder is obtained after #j and #k are rounded to their nearest whole numbers. When #j is a negative value, #i is assumed to be a negative value.)
<4> Functions	#i=SIN[#j]	Sine (in degrees)
	#i=COS[#j]	Cosine (in degrees)
	#i=TAN[#j]	Tangent (in degrees)
	#i=ASIN[#j]	Arc sine
	#i=ACOS[#j]	Arc cosine
	#i=ATAN[#j]	Arc tangent (one argument), ATN can also be used.
	#i=ATAN[#j]/[#k]	Arc tangent (two arguments), ATN can also be used.
	#i=ATAN[#j,#k]	Arc tangent (two arguments), ATN can also be used.
	#i=SQRT[#j]	Square root, SQR can also be used.
	#i=ABS[#j]	Absolute value
	#i=BIN[#j]	Conversion from BCD to binary
	#i=BCD[#j]	Conversion from binary to BCD
	#i=ROUND[#j]	Rounding off, RND can also be used.
	#i=FIX[#j]	Rounding down to an integer
	#i=FUP[#j]	Rounding up to an integer
	#i=LN[#j]	Natural logarithm
	#i=EXP[#j]	Exponent using base e (2.718...)
	#i=POW[#j,#k]	Power (#j to the #kth power)
	#i=ADP[#j]	Addition of a decimal point
	#i=PRM[#j]	Parameter reading (system common, path, or machine group parameter)
	#i=PRM[#j,#k]	Parameter reading (system common, path, or machine group parameter bit number specification)
	#i=PRM[#j]/[#l]	Parameter reading (axis or spindle parameter)
	#i=PRM[#j,#k]/[#l]	Parameter reading (axis or spindle parameter bit number specification)

Explanation

- Angle units

The units of angles used with the SIN, COS, ASIN, ACOS, TAN, and ATAN functions are degrees. For example, 90 degrees and 30 minutes is represented as 90.5 degrees.

- ARCSIN #i = ASIN[#j];

- The solution ranges are as indicated below:
  When the bit 0 (NAT) of parameter No. 6004 is set to 0: 270° to 90°
  When the bit 0 (NAT) of parameter No. 6004 is set to 1: -90° to 90°
- When #j is beyond the range of -1 to 1, an alarm PS0119, "ARGUMENT VALUE OUT OF RANGE" is issued.
- A constant can be used instead of the #j variable.

- ARCCOS #i = ACOS[#j];

- The solution ranges from 180° to 0°.
- When #j is beyond the range of -1 to 1, an alarm PS0119, "ARGUMENT VALUE OUT OF RANGE" is issued.
- A constant can be used instead of the #j variable.

- ARCTAN #i = ATAN[#j]/[#k]; (two arguments)

- ATAN[#j,#k] is equivalent to ATAN[#j]/[#k].
- When point (#k,#j) on plane X-Y is given, this function returns the value of the arc tangent for the angle made by the point.
- A constant can be used instead of the #j variable.
- The solution ranges are as follows:
  When the bit 0 (NAT) of parameter No. 6004 is set to 0: 0° to 360°
  Example: When #1 = ATAN[-1]/[-1]; is specified, #1 is 225.0.
  When the bit 0 (NAT) of parameter No. 6004 is set to 1: -180° to 180°
  Example: When #1 = ATAN[-1]/[-1]; is specified, #1 is -135.0.

- ARCTAN #i = ATAN[#j]; (one argument)

- When ATAN is specified with one argument, this function returns the main value of arc tangent (-90° ≤ ATAN[#j] ≤ 90°). In other word, this function returns the same value as ATAN in calculator specifications.
- To use this function as the dividend of a division, be sure to enclose it with brackets ([]). If this function is not enclosed, ATAN[#j]/[#k] is assumed.
  Example:
  #100 = [ATAN[1]]/10 ; : Divides ATAN with one argument by 10.
  #100 = ATAN[1]/[10] ; : Executes ATAN with two arguments.
  #100 = ATAN[1]/10 ; : Assumes ATAN with two arguments, but issues an alarm PS1131, "MISSING OPEN BRACKET" because the X coordinate specification is not enclosed with brackets ([]).

- Natural logarithm #i = LN[#j];

- When the antilogarithm (#j) is zero or smaller, an alarm PS0119, "ARGUMENT VALUE OUT OF RANGE" is issued.
- A constant can be used instead of the #j variable.

- Exponential function #i = EXP[#j];

- When the result of the operation overflows, an alarm PS0119, "ARGUMENT VALUE OUT OF RANGE" is issued.
- A constant can be used instead of the #j variable.

- ROUND function

- When the ROUND function is included in an arithmetic or logic operation command, IF statement, or WHILE statement, the ROUND function rounds off at the first decimal place.
  Example:
  When #1=ROUND[#2]; is executed where #2 holds 1.2345, the value of variable #1 is 1.0.
- When the ROUND function is used in NC statement addresses, the ROUND function rounds off the specified value according to the least input increment of the address.
  Example:
  Creation of a drilling program that cuts according to the values of variables #1 and #2, then returns to the original position
  Suppose that the increment system is 1/1000 mm, variable #1 holds 1.2345, and variable #2 holds 2.3456. Then,
  G00 G91 X-#1; Moves 1.235 mm in negative direction.
  G01 X-#2 F300; Moves 2.346 mm in negative direction.
  G00 X[#1+#2]; Since 1.2345 + 2.3456 = 3.5801 in positive direction, the travel distance is 3.580, which does not return the tool to the original position.
  This difference comes from whether addition is performed before or after rounding off. G00X-[ROUND[#1]+ROUND[#2]]; must be specified to return the tool to the original position.

- Add decimal point (ADP) function

- ADP[#n] (n = 1 to 33) can be executed to add a decimal point to an argument passed with no decimal point, in the subprogram.
  Example:
  In the subprogram called with G65 P_X10;, the value of ADP[#24] is a value to which a decimal point is added at its end (that is, 10.). Use this function when you do not want to consider the increment system in the subprogram. When bit 4 (CVA) of parameter No. 6007 is set to 1, however, the ADP function cannot be used because any argument is converted to 0.01 the moment it is passed.

NOTE
For compatibility among programs, it is recommended that the ADP function be not used, and decimal points be added in the argument specification for a macro call.

- Rounding up and down to an integer (FUP and FIX)

With CNC, when the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. Conversely, when the absolute value of the integer produced by an operation on a number is less than the absolute value of the original number, such an operation is referred to as rounding down to an integer. Be particularly careful when handling negative numbers.
Example:
Suppose that #1=1.2 and #2=-1.2.
When #3=FUP[#1] is executed, 2.0 is assigned to #3.
When #3=FIX[#1] is executed, 1.0 is assigned to #3.
When #3=FUP[#2] is executed, -2.0 is assigned to #3.
When #3=FIX[#2] is executed, -1.0 is assigned to #3.

- Parameter reading (PRM)

- The following formats are available depending on the parameter type and the data to be read.

Format	Remarks
#i = PRM[#j];	Format for a system common, path, or machine group parameter.
#i = PRM[#j, #k];	Format for specifying the number of a bit of a system common, path, or machine group parameter.
#i = PRM[#j] /[#l];	Format for an axis or spindle parameter
#i =PRM[#j, #k] /[#l];	Format for specifying the number of a bit of an axis or spindle parameter

- For #j, set a parameter number.
- For #k, set a bit number.
- For #l, set an axis number or a spindle number.

・ By specifying a path number, it is possible to read a parameter for that path. There are two ways to specify a path number.
  a) Adding a path number to a parameter number
  b) Specifying a path number using a system variable

  a) Adding a path number to a parameter number
  By adding a path number to the 8th and 9th high-order digits of the parameter number specified with the operation command PRM, it is possible to read the parameter for the specified path.
  #i = PRM[#j];
  #j = ppxxxxxxx ; For pp, set a path number and for xxxxxxx, set a parameter number.

NOTE
1 If a path number is omitted or if 0 is specified, a parameter for the local path will be read.
2 If a path number out of the range is specified, alarm PS0115, “VARIABLE NO. OUT OF RANGE” is issued.
3 To read and write data for the 10th path, set bit 0 (F16) of parameter No. 6008 to 0.
If parameter bit F16 is 1, the calculation precision is up to the 8th digit. Thus, if a parameter number is determined with a calculation command such as #100= PRM[100000000 +#1] when the data for the 10th path is read or written, the correct number may not be determined.

  b) Specifying a path number using a system variable
  By specifying a path number using system variable #3018, it is possible to read a parameter for the specified path.

Example
Reading the fourth axis of parameter No. 01322 for the second path
```
#3018 = 2 ; ..... Set the second path.
#1 =PRM[1322] /[4] ; .. Read the fourth axis of parameter No. 01322 for the
    :                   second path and write it to variable #1 for the local path.
#3018= 0 ; ..... Return the path number to that of the local path.
```

- Abbreviations of arithmetic and logic operation commands

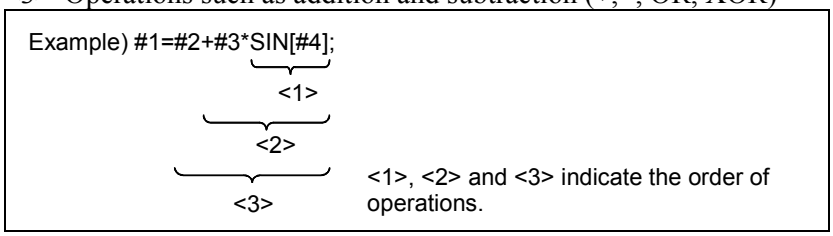
When a function is specified in a program, the first two characters of the function name can be used to specify the function.

Example: ROUND → RO
FIX → FI

NOTE
POW cannot be abbreviated.

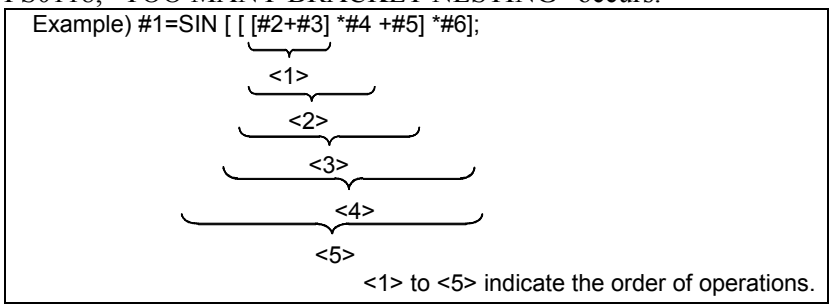
- Priority of operations

<1> Functions
<2> Operations such as multiplication and division (*, /, AND)
<3> Operations such as addition and subtraction (+, -, OR, XOR)



- Bracket nesting

Brackets are used to change the order of operations. Brackets can be used to a depth of five levels including the brackets used to enclose a function. When a depth of five levels is exceeded, an alarm PS0118, "TOO MANY BRACKET NESTING" occurs.



Limitation

- **Caution concerning decreased precision**
  When bit 0 (F16) of parameter No. 6008 is set to 0

- Addition and subtraction

  Note that when an absolute value is subtracted from another absolute value in addition or subtraction, the relative error may become 10^{-15} or greater.

  For example, assume that #1 and #2 have the following true values in the process of operation.

  (The following values are examples in the process of operation and cannot actually be specified from any program.)

  #1=9876543210.987654321
  #2=9876543210.987657777

  You cannot obtain the following result with operation #2-#1:

  #2-#1=0.000003456

  This is because the precision of custom macro variables is 15 decimal digits. With this precision, the values of #1 and #2 become:

  #1=9876543210.987650000
  #2=9876543210.987660000

  (Precisely, the actual values are slightly different from the above values because they are internally processed in binary.) Therefore, the result is:

  #2-1=0.000010000

  A large error occurs.

- Logical expressions

  Be aware of errors that can result from conditional expressions using EQ, NE, GT, LT, GE, and LE because they are processed basically in the same way as addition and subtraction. For example, if the following statement is used to decide whether #1 is equal to #2 in the above example, a correct decision may not be resulted because errors may occur:

  IF [#1 EQ #2]

Evaluate the difference between #1 and #2 with:

IF [ABS [#1-#2]LT 0.1]

Then, assume that the values are equal when the difference does not exceed the allowable error range.

- Trigonometric functions
  The absolute error is guaranteed for trigonometric functions. However, the relative error is 10^{-15} or greater. Carefully perform multiplication or division after executing a trigonometric function.
- FIX function
  When using the FIX function for the result of an operation, be careful with the precision. For example, when the following operations are performed, the value of #3 may not always be 2.

```
N10 #1=0.002;
N20 #2=#1*1000;
N30 #3=FIX[#2];
```

This is because an error may occur in operation N20 and the result may not be

#2=2.0000000000000000

but a value a little smaller than 2 such as the following:

#2=1.9999999999999997

To prevent this, specify N30 as follows:

N30 #3=FIX[#2+0.001];

Generally, specify the FIX function as follows:

FIX[expression] → FIX[expression ±ε]

(Specify +ε when the value of the expression is positive or -ε when it is negative, and 0.1, 0.01, 0.001, ... for ε as required.)

NOTE
The operation result of exponential function #i=EXP[#j]; overflows when #j exceeds about 790.

When bit 0 (F16) of parameter No. 6008 is set to 1

Errors may occur when operations are performed.

Table 16.4 (b) Errors involved in operations

Operation	Average error	Maximum error	Type of error
a = b*c	1.55×10^{-10}	4.66×10^{-10}	Relative error(*1) $\left\lvert \frac{\varepsilon}{a} \right\rvert$
a = b / c	4.66×10^{-10}	1.88×10^{-9}	
a = $\sqrt{b}$	1.24×10^{-9}	3.73×10^{-9}	
a = b + c a = b – c	2.33×10^{-10}	5.32×10^{-10}	(*2) MIN $\left\lvert \frac{\varepsilon}{b} \right\rvert$ $\left\lvert \frac{\varepsilon}{c} \right\rvert$
a = SIN [b] a = COS [b]	5.0×10^{-9}	1.0×10^{-8}	Absolute error(*3) $\lvert \varepsilon \rvert$ degrees
a = ATAN [b] / [c]	1.8×10^{-6}	3.6×10^{-6}	

NOTE
1 The relative error depends on the result of the operation.
2 Smaller of the two types of errors is used.
3 The absolute error is constant, regardless of the result of the operation.
4 Function TAN performs SIN/COS.
5 Note that, in the case of natural logarithm #i=LN[#j]; and exponential function #i=EXP[#j];, the relative error may become 10^{-8} or greater.
6 The operation result of exponential function #i=EXP[#j]; overflows when #j exceeds about 110.

- The precision of variable values is about 8 decimal digits. When very large numbers are handled in an addition or subtraction, the expected results may not be obtained.
  Example:
  When an attempt is made to assign the following values to variables #1 and #2:
  #1=9876543210123.456
  #2=9876543277777.777
  the values of the variables become:
  #1=9876543200000.000
  #2=9876543300000.000
  In this case, when #3=#2-#1; is calculated, #3=100000.000 results. (The actual result of this calculation is slightly different because it is performed in binary.)
- Also be aware of errors that can result from conditional expressions using EQ, NE, GE, GT, LE, and LT.
  Example:
  IF[#1 EQ #2] is effected by errors in both #1 and #2, possibly resulting in an incorrect decision.
  Therefore, instead find the difference between the two variables with IF[ABS[#1-#2] LT 0.001].
  Then, assume that the values of the two variables are equal when the difference does not exceed an allowable limit (0.001 in this case).
- Also, be careful when rounding down a value.
  Example:
  When #2=#1*1000; is calculated where #1=0.002;, the resulting value of variable #2 is not exactly 2 but 1.99999997.
  Here, when #3=FIX[#2]; is specified, the resulting value of variable #3 is not 2.0 but 1.0.
  In this case, round down the value after correcting the error so that the result is greater than the expected number, or round it off as follows:
  #3=FIX[#2+0.001]
  #3=ROUND[#2]

- Brackets

Brackets ([ ]) are used to enclose an expression.
Note that parentheses ( ) are used for comments.

- Divisor

When a divisor of zero is specified in a division, an alarm PS0112, “ZERO DIVIDE” occurs.

16.5 INDIRECT AXIS ADDRESS SPECIFICATION

Overview

When the custom macro function is enabled, you can use AX[(axis-number)] in an axis address specification to indirectly specify an axis with its axis number and not to directly specify it with its axis name.
You can also use AXNUM[(axis-name)] to obtain the axis number for an axis name.

Explanation

- Indirect axis address

You can use indirect axis address AX[ ] to specify an axis with its axis number. (An equal sign (=) is always required following AX[ ].)

AX[(axis-number)] = (numeric-value) ;

(axis-number): 1 to the number of controlled axes (number of controlled axes for each path in a multi-path system)
(numeric-value): Value specified for the axis specified with its axis number

If an invalid axis number is specified, an alarm PS0331, "ILLEGAL AXIS NUMBER IN AX[]" occurs.
If a value is put in a decimal place, the number is rounded off to an integer and the result is treated as the axis number.
You can also specify a variable (local, common, or system variable) for (axis-number). To specify an operation using a variable name in (axis-number), however, enclose the variable name with brackets ([ ]).

1. **AX[1]=100.0;**
   Specifies a value of 100.000 for the first axis.
2. **AX[#500]=200.0;**
   Specifies a value of 200.000 for the axis having the axis number indicated by the value stored in #500.
3. **AX[#500+1]=300.0;**
   Specifies a value of 300.000 for the axis having the axis number obtained by adding 1 to the value stored in #500.
4. **SETVN 500 [ABC];**
   AX[#ABC]=400.0;
   Specifies a value of 400.000 for the axis having the axis number indicated by the value stored in #ABC (#500).
5. **SETVN 500 [ABC];**
   AX[[#ABC]+1]=500.0;
   Specifies a value of 400.000 for the axis having the axis number obtained by adding 1 to the value stored in #ABC (#500).
6. **SETVN 500 [ABC];**
   AX[#ABC+1]=500.0;
   Alarm PS0331 occurs.

- AXNUM function

You can use AXNUM[ ] to obtain an axis number.

AXNUM[(axis-name)];
If an invalid axis name is specified, an alarm PS0332, "ILLEGAL AXIS ADDRESS IN AXNUM[]" occurs.

When the number of controlled axes is 3, the name of the first axis is X, that of the second axis is Y, and that of the third axis is Z

1. **#500=AXNUM[X];**
   A value of 1 is stored in #500.
2. **#501=AXNUM[Y];**
   A value of 2 is stored in #501.
3. **#502=AXNUM[Z];**
   A value of 3 is stored in #502.
4. **#503=AXNUM[A];**
   Alarm PS0332 occurs.

Example

Examples when the name of the first axis is X, that of the second axis is Y, and that of the third axis is Z1

```
N10 SETVN 500[AXIS1,AXIS2,AXIS3] ;
N20 [#AXIS1]=AXNUM[X] ;
N30 [#AXIS2]=AXNUM[Y] ;
N40 [#AXIS3]=AXNUM[Z1] ;
N50 G92 AX[#AXIS1]=0 AX[#AXIS2]=0 AX[#AXIS3]=0 ;
N60 G90G01F1000. ;
N70 AX[#AXIS1]=100.0 AX[#AXIS2]=100.0 AX[#AXIS3]=100.0 ;
N80 G02 AX[#AXIS1]=100. 0 AX[#AXIS2]= 0.0 R50.0 ;
N90 M02;
```

Limitation

When the custom macro function is enabled, AX and AXN cannot be used as an extended axis name. AX is assumed to be AX[ ] and AXN is assumed to be AXNUM[ ].

16.6 READING PARAMETERS

Overview

By using the PRM function, it is possible to read parameters.

Format	Remarks
#i = PRM[#j, #k] ;	In the case of parameters other than axis type parameters
#i = PRM[#j, #k] / [#l] ;	In the case of axis type parameters

Explanation

- Reading a parameter
#i=PRM[#j,#k]
#i=PRM[#j,#k]/[#l]

- For #j, enter a parameter number. If the number of a parameter that cannot be read, alarm PS0119, "ARGUMENT VALUE OUT OF RANGE", is issued.
- To read a bit type parameter, specify, for #k, the bit number of the bit type parameter in the range of 0 to 7. If a bit number is specified, data with the specified bit is read. If none is specified, data with all bits is read. For parameters other than bit type parameters, the bit number is ignored.
- For #l, set the axis number of an axis type parameter in the range of 1 to 8. If an axis type parameter is to be read but #l is not specified, alarm PS0119 is issued. For parameters other than axis type parameters, #1 may be omitted together with '/'.

Example

1. Reading the value of the third axis of bit 0 (MIR) of bit axis type parameter No. 0012
   If parameter No. 0012 (third axis) = 10010001
   #2=12 ; Parameter number setting
   #3=0 ; Bit number setting
   #4=3 ; Axis number setting
   If reading data with all bits
   #1=PRM[#2]/[#4] ; #1=10010001
   If reading data with a specified bit
   #1=PRM[#2, #3]/[#4] ; #1=1

2. Reading the value of the fourth axis of axis type parameter No. 1322
   #2=1322 ; Parameter number setting
   #4=4 ; Axis number setting
   #1=PRM[#2]/[#4] ;

3. Reading bit 2 (SBP) of bit type parameter No. 3404
   If parameter No. 0012 (third axis) = 01010000
   #2=3404 ; Parameter number setting
   #3=2 ; Bit number setting
   If reading data with all bits
   #1=PRM[#2] ; #1=1010000
   If reading data with a specified bit
   #1=PRM[#2,#3] ; #1=0

16.7 MACRO STATEMENTS AND NC STATEMENTS

The following blocks are referred to as macro statements:

- Blocks containing an arithmetic or logic operation (=)
- Blocks containing a control statement (such as GOTO, DO, END)
- Blocks containing a macro call command (such as macro calls by G65, G66, G66.1, G67, or other G codes, or by M codes)

Any block other than a macro statement is referred to as an NC statement.

Explanation

- Differences from NC statements

- Even when single block mode is on, the machine does not stop. Note, however, that the machine stops in the single block mode when bit 5 of parameter SBM No. 6000 is 1.
- Macro blocks are not regarded as blocks that involve no movement in the cutter compensation mode.

- NC statements that have the same property as macro statements

- NC statements that include a subprogram call command (such as subprogram calls by M98 or other M codes, or by T codes) and not include other command addresses except an O, file name, N, P, or L address have the same property as macro statements.
- NC statements that include M99 and those do not include other command addresses except an O, file name, N, P, or L address have the same property as macro statements.

16.8 BRANCH AND REPETITION

In a program, the flow of control can be changed using the GOTO statement and IF statement. Three types of branch and repetition operations are used:

Branch and repetition	GOTO	(unconditional branch)
	IF	(conditional branch: if ..., then...)
	WHILE	(repetition while ...)

16.8.1 Unconditional Branch (GOTO Statement)

A branch to sequence number n occurs. When a sequence number outside of the range 1 to 99999999 is specified, an alarm PS1128, “SEQUENCE NUMBER OUT OF RANGE” occurs. A sequence number can also be specified using an expression.

GOTOn ; n: Sequence number (1 to 99999999)

Example: GOTO 1;
GOTO #10;

⚠ WARNING
Do not specify multiple blocks with the same sequence number in a single program. It is very dangerous to specify such blocks because the destination of a branch from the GOTO statement is undefined.

NOTE
1 A backward branch takes more time as compared with a forward branch.
2 In the destination of GOTOn, that is, the block with sequence number n, the sequence number must appear at the beginning of the block. If the sequence number is not at the beginning of the block, a branch cannot be made.

16.8.2 GOTO Statement Using Stored Sequence Numbers

When the GOTO statement is executed in a custom macro control command, a sequence number search is made for sequence numbers stored at previous execution of the corresponding blocks at a high speed.
As a "sequence number stored at previous execution of the corresponding block", a unique sequence number within the same program or the sequence number of a subprogram call with which the block was executed is stored.
The storage type differs depending on the values of the following parameters.

(1) When bit 1 (MGO) of parameter No. 6000 is set to 1
- Fixed type: Up to 20 sequence numbers stored at execution of the corresponding blocks from the start of operation

(2) When bit 4 (HGO) of parameter No. 6000 is set to 1
- Variable type: Up to 30 sequence numbers stored at execution of the corresponding blocks before execution of the GOTO statement
- History type: Up to 10 sequence numbers stored by a sequence number search previously made using the GOTO statement

The stored sequence numbers are canceled in the following cases:
- Immediately after power-on
- After a reset
- Operation after program registration or editing (including background editing and MDI program editing)

⚠ WARNING

Do not specify multiple blocks with the same sequence number in a single program.

It is very dangerous to specify the sequence number of the branch destination before and after the GOTO statement and execute the GOTO statement because the branch destination changes according to the values of the parameters as shown below:

When bit 1 (MGO) or 4 (HGO) of parameter No. 6000 is set to 1	When both bits 1 (MGO) and 4 (HGO) of parameter No. 6000 are set to 0
: N10; : GOTO10; : N10; A branch to N10 before the GOTO statement occurs.	: N10; : GOTO10; : N10; A branch to N10 after the GOTO statement occurs.

When bit 1 (MGO) or 4 (HGO) of parameter No. 6000 is set to 1 and the GOTO statement is executed, the sequence number of the branch destination may not be contained in the sequence numbers stored at previous execution of the corresponding blocks. In this case, a branch to the sequence number in a block following the GOTO statement occurs (the destination is the same as when both bits are set to 0).

NOTE

When an external program is read and executed by DNC operation, the executed sequence numbers are not stored.

When a program registered in memory is executed by a subprogram call, the sequence numbers are stored.

⚠ CAUTION

According to the restrictions on the GOTO statement, no branch to a sequence number within a DO-END loop cannot be made. If a program in which a branch to a sequence number within a loop occurs is executed, operation may differ depending on whether the GOTO statement using stored sequence numbers is used.

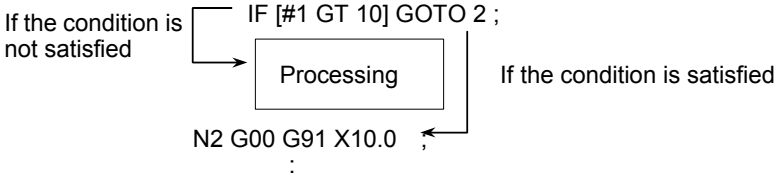
16.8.3 Conditional Branch (IF Statement)

Specify a <conditional expression> after IF.

IF[<conditional expression>]GOTOn

If the specified <conditional expression> is satisfied (true), a branch to sequence number n occurs. If the specified condition is not satisfied, the next block is executed.

If the value of variable #1 is greater than 10, a branch to sequence number N2 occurs.



IF[<conditional expression>]THEN

If the specified <conditional expression> is satisfied (true), a macro statement specified after THEN is executed.
Only a single macro statement is executed.

```
If the values of #1 and #2 are the same, 0 is assigned to #3.
IF[#1 EQ #2] THEN#3=0 ;
```

```
If the values of #1 and #2 are the same and those of #3 and #4 are also the same, 0 is assigned to #5.
IF[[#1 EQ #2] AND [#3 EQ #4]] THEN#5=0 ;
```

```
If the values of #1 and #2 are the same or those of #3 and #4 are the same, 0 is assigned to #5.
IF[[#1 EQ #2] OR [#3 EQ #4]] THEN#5=0 ;
```

Explanation

- <Conditional expression>

<Conditional expressions> are divided into <simple conditional expressions> and <complex conditional expressions>. In a <simple conditional expression>, a relational operator described in Fig. 16.8.3 (a) is specified between two variables or between a variable and constant to be compared. An <expression> can be used instead of a variable. With a <complex conditional expression>, an AND (logical AND), OR (logical OR), or XOR (exclusive OR) operation is performed for the results (true or false) of multiple <simple conditional expressions>.

- Relational operators

Relational operators each consist of two letters and are used to compare two values to determine whether they are equal or one value is smaller or greater than the other value. Note that the equal sign (=) and inequality sign (>, <) cannot be used as a relational operator.

Fig. 16.8.3 (a) Relational operators

Operator	Meaning
EQ	Equal to(=)
NE	Not equal to(≠)
GT	Greater than(>)
GE	Greater than or equal to(≥)
LT	Less than(<)
LE	Less than or equal to(≤)

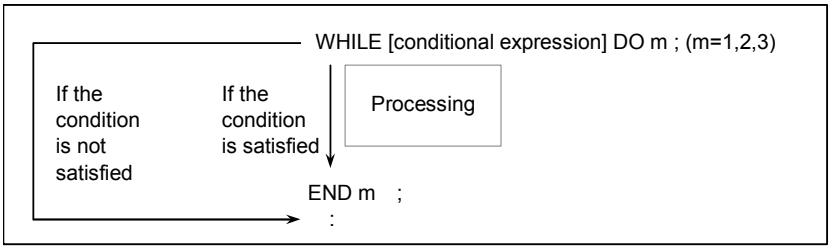
Sample program

The sample program below finds the total of numbers 1 to 10.

```
O9500;
  #1=0;..... Initial value of the variable to hold the sum
  #2=1;..... Initial value of the variable as an addend
N1 IF[#2 GT 10] GOTO 2;... Branch to N2 when the addend is greater than 10
  #1=#1+#2;..... Calculation to find the sum
  #2=#2+1;..... Next addend
  GOTO 1; ..... Branch to N1
N2 M30;..... End of program
```

16.8.4 Repetition (WHILE Statement)

Specify a conditional expression after WHILE. While the specified condition is satisfied, the program from DO to END is executed. If the specified condition is not satisfied, program execution proceeds to the block after END.

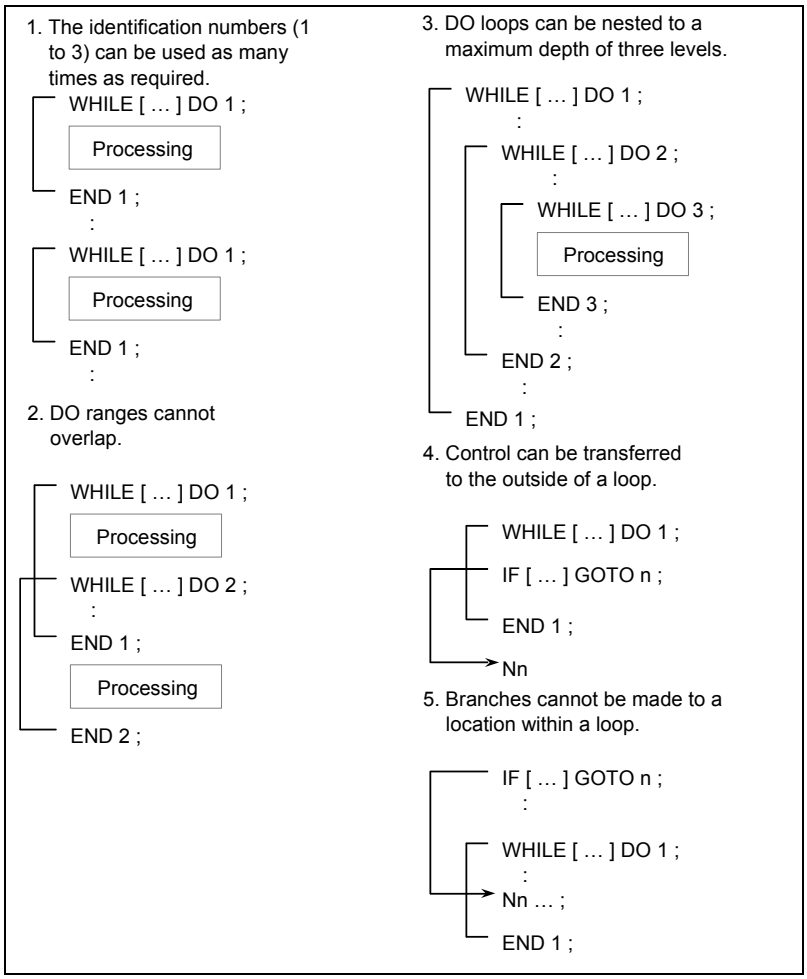


Explanation

While the specified condition is satisfied, the program from DO to END after WHILE is executed. If the specified condition is not satisfied, program execution proceeds to the block after END. The same format as for the IF statement applies. A number after DO and a number after END are identification numbers for specifying the range of execution. The numbers 1, 2, and 3 can be used. When a number other than 1, 2, and 3 is used, an alarm PS0126, “ILLEGAL LOOP NUMBER” occurs.

- Nesting

The identification numbers (1 to 3) in a DO-END loop can be used as many times as desired. Note, however, when a program includes crossing repetition loops (overlapped DO ranges), an alarm PS0124, “MISSING END STATEMENT” occurs.



Limitation

- Infinite loops

When DO m is specified without specifying the WHILE statement, an infinite loop ranging from DO to END is produced.

- Processing time

When a branch to the sequence number specified in a GOTO statement occurs, the sequence number is searched for. For this reason, processing in the reverse direction takes a longer time than processing in the forward direction. Therefore, in the case of processing in the reverse direction, use the WHILE statement for repetition to reduce processing time.

- Undefined variable

In a conditional expression that uses EQ or NE, a <null> and zero have different effects. In other types of conditional expressions, a <null> is regarded as zero.

Sample program

The sample program below finds the total of numbers 1 to 10.

```
O0001;
#1=0;
#2=1;
WHILE[#2 LE 10]DO 1;
#1=#1+#2;
#2=#2+1;
END 1;
M30;
```

16.8.5 Precision Specification for Macro Relational Operators

When making a comparison using a relational operator in a custom macro, you can specify the number of decimal places in the two values to be compared. The values are compared after rounded off to the specified number of decimal places.

Setting bit 5 (EDP) of parameter No. 6019 to 1 enables this function. Specify the number of significant decimal places you want to set in parameter No. 6100.

Relational operators

The precision specification is valid for the following relational operators.

Operator	Meaning
EQ	Equal to(=)
NE	Not equal to(≠)
GT	Greater than(>)
GE	Greater than or equal to(≥)
LT	Less than(<)
LE	Less than or equal to(≤)

Limitation

The precision specification for macro relational operators is valid for the following functions:

- Custom macro
- Macro executor

NOTE
The precision specification is invalid for real-time custom macros.

Example

Macro program

```
O1000 ;
#100=1.2344567 ;
#101=1.2345678 ;
N10 IF[#100 EQ #101] GOTO 20 ;
...
N20 ;
N30 IF[#100 NE #101] GOTO 40 ;
...
N40 ;
...
```

When a comparison is executed, the target two values are compared after rounded off to the specified number of decimal places.
The actual values are not rounded.

#100=1.2344567
ˆˆˆ The number of significant decimal places is 3.
ˆ Rounded off.

1. When the precision specification for macro relational operators is disabled
   Bit 5 (EDP) of parameter No. 6019 is set to 0.
   The values of #100 and #101 are compared as is. In the N10 block, a branch to N20 does not occur. In the N30 block, a branch to N40 occurs.

2. When the precision specification for macro relational operators is enabled - 1
   Bit 5 (EDP) of parameter No. 6019 is set to 1 and parameter No. No.6100 is set to 2.
   The value obtained by rounding off the value of #100, 1.23, is compared with the value obtained by rounding off the value of #101, 1.23. The values of #100 and #101 are not changed.
   In the N10 block, a branch to N20 occurs. In the N30 block, a branch to N40 does not occur.

3. When the precision specification for macro relational operators is enabled - 2
   Bit 5 (EDP) of parameter No. 6019 is set to 1 and parameter No. No.6100 is set to 3.
   The value obtained by rounding off the value of #100, 1.234, is compared with the value obtained by rounding off the value of #101, 1.235. The values of #100 and #101 are not changed.
   In the N10 block, a branch to N20 does not occur. In the N30 block, a branch to N40 occurs.

When variable values are negative, processing is performed as follows:

Macro program

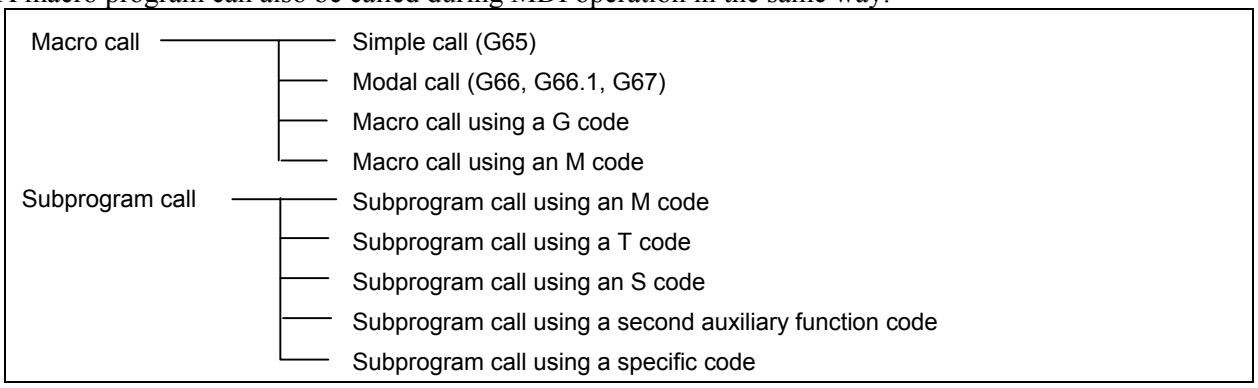
```
O1000 ;
#100=-1.2344 ;
#101=-1.2345 ;
N10 IF[#100 EQ #101] GOTO 20 ;
...
N20 ;
...
```

#100=-1.2345
ˆˆˆ The number of significant decimal places is 3.
ˆ Rounded off.

1. When the precision specification for macro relational operators is enabled
   Bit 5 (EDP) of parameter No. 6019 is set to 1 and parameter No. No.6100 is set to 3.
   The value obtained by rounding off the value of #100, -1.234, is compared with the value obtained by rounding off the value of #101, -1.235. The values of #100 and #101 are not changed.
   In the N10 block, a branch to N20 does not occur.

16.9 MACRO CALL

A macro program can be called using the following methods. The calling methods can roughly be divided into two types: macro calls and subprogram calls.
A macro program can also be called during MDI operation in the same way.



Limitation

- Call nesting

Macro calls can be nested to a depth of up to five levels and subprogram calls can be nested to a depth of up to ten levels; calls can be nested to a depth of up to 15 levels in total.

- Differences between macro calls and subprogram calls

Macro call (G66, G66.1, Ggg, or Mmm) differs from subprogram call (for example M98, Mmm, or Ttt) as described below.

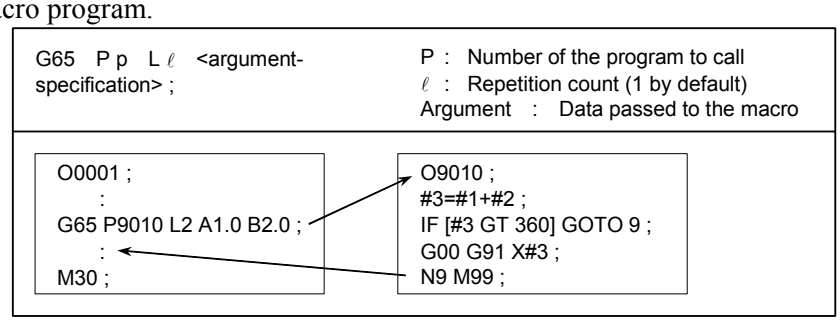
- With a macro call, an argument (data passed to a macro) can be specified. A subprogram call does not have this capability.
- If a macro call block contains another NC command (such as G01 X100.0 G65 Pp), an alarm PS0127, “DUPLICATE NC,MACRO STATEMENT” occurs.
- If a subprogram call block contains another NC command (such as G01 X100.0 M98 Pp), the subprogram is called after the command is executed.
- In any macro call block, the machine does not stop in the single block mode.
  If a subprogram call block contains another NC command (such as G01 X100.0 M98 Pp), the machine stops in the single block mode.
- With a macro call, the level of local variables changes. With a subprogram call, the level of local variables does not change. (See "Local variable levels" in Limitation of Subsection, “Simple Call (G65)”.)

- Called program and folders to be searched

The order in which folders are called depends on the method of calling a macro or subprogram.
Folders are searched in sequence and the program found first is called. For details, see the "Managing Programs" chapter.

16.9.1 Simple Call (G65)

When G65 is specified, the custom macro specified at address P is called. Data (argument) can be passed to the custom macro program.



Explanation

- Call

- After G65, specify at address P the program number of the custom macro to call.
- When a number of repetitions is required, specify a number from 1 to 999999999 after address L. When L is omitted, 1 is assumed.
- By using argument specification, values are assigned to corresponding local variables.

- Argument specification

Two types of argument specification are available. Argument specification I uses letters other than G, L, O, N, and P once each. Argument specification II uses A, B, and C once each and also uses I, J, and K up to ten times. The type of argument specification is determined automatically according to the letters used.

- Argument specification I

Address	Variable number
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11

Address	Variable number
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
S	#19

Address	Variable number
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

- Addresses G, L, N, O, and P cannot be used in arguments.
- Addresses that need not be specified can be omitted. Local variables corresponding to an omitted address are set to null.
- Addresses do not need to be specified alphabetically. They conform to word address format.
  I, J, and K need to be specified alphabetically, however.
  Argument specification I is always used for I, J, and K by setting bit 7 (IJK) of parameter No. 6008 to 1

Example

- When bit 7 (IJK) of parameter No. 6008 is 0, I_J_K_ means that I = #4, J = #5, and K = #6 while K_J_I_ means K = #6, J = #8, and I= #10 because argument specification II is used.
- When bit 7 (IJK) of parameter No. 6008 is 1, K_J_I_ means that I = #4, J = #5, and K = #6, which is the same as with I_J_K_, because argument specification I is used.

- Argument specification II
  Argument specification II uses A, B, and C once each and uses I, J, and K up to ten times. Argument specification II is used to pass values such as 3-dimensional coordinates as arguments.

Address	Variable number
A	#1
B	#2
C	#3
I_1	#4
J_1	#5
K_1	#6
I_2	#7
J_2	#8
K_2	#9
I_3	#10
J_3	#11

Address	Variable number
IK_3	#12
I_4	#13
J_4	#14
K_4	#15
I_5	#16
J_5	#17
K_5	#18
I_6	#19
J_6	#20
K_6	#21
I_7	#22

Address	Variable number
J_7	#23
K_7	#24
I_8	#25
J_8	#26
K_8	#27
I_9	#28
J_9	#29
K_9	#30
I_{10}	#31
J_{10}	#32
K_{10}	#33

- Subscripts of I, J, and K for indicating the order of argument specification are not written in the actual program.

NOTE
When bit 7 (IJK) of parameter No. 6008 is 1, argument II cannot be used.

Limitation

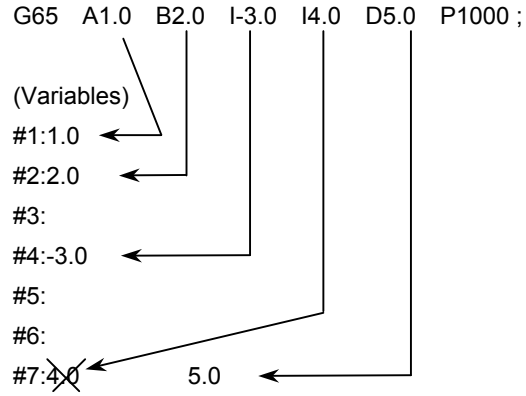
- Format

G65 must be specified before any argument.

- Mixture of argument specifications I and II

The CNC internally identifies argument specification I and argument specification II. If a mixture of argument specification I and argument specification II is specified, the type of argument specification specified later takes precedence.

[Example]



When both the I4.0 and D5.0 arguments are commanded for variable #7 in this example, the latter, D5.0, is valid.

- Position of the decimal point

The units used for argument data passed without a decimal point correspond to the least input increment of each address.

⚠ CAUTION
The value of an argument passed without a decimal point may vary according to the system configuration of the machine. It is good practice to use decimal points in macro call arguments to maintain program compatibility.

- Extended axis name

The axis address of an extended axis name cannot be specified as an argument. If an attempt is made to specify it, alarm PS0129, "'G' AS ARGUMENT" is issued.

M

When a value is specified with no decimal point, the number of decimal places is determined as follows.

Address	For a non-axis address	For an axis address
D, E, H, M, S, or T	0	
Q or R	α (NOTE 2)	
A, C, I, J, K, X, Y, or Z	α (NOTE 2)	β (NOTE 3)
B, U, V(NOTE 1), or W	0	β (NOTE 3)
Second auxiliary function	γ (NOTE 4)	

Address	Metric input	Inch input
F (G93 mode)	3	
F (G94 mode)	0	2
F (G95 mode)	2 (NOTE 5)	4 (NOTE 5)

NOTE

1 When V is used in a call using a specific code, the number of decimal places is determined according to the setting for the reference axis.

2 α is determined according to the increment system for the reference axis (axis specified with parameter No. 1031) as listed in the table in NOTE 3.

3 β is determined according to the increment system for the corresponding axis address as listed in the following table.

Increment system	Linear axis (metric input)	Linear axis (inch input)	Rotation axis
IS-A	2	3	2
IS-B	3	4	3
IS-C	4	5	4
IS-D	5	6	5
IS-E	6	7	6

When bit 7 (IPR) of parameter No. 1004 is set to 1, the above values from which 1 is subtracted are used. When the increment system for an axis is IS-A, however, the setting of bit 7 (IPR) of parameter No. 1004 is not effective.
When calculator-type decimal notation for each axis is used (bit 0 (ADX) of parameter No. 3455 is set to 1), the number of decimal places is 0. When bit 7 (EAP) of parameter No. 3452 is set to 1, however, calculator-type decimal notation is not effective and the number of decimal places is determined as listed in the above table.

4 γ is determined according to the increment system for the reference axis (axis specified with parameter No. 1031) as listed in the following table. (When bit 7 (BDX) of parameter No. 3450 is set to 1, γ is also determined in the same way.)

Increment system for the reference axis	AUP (No.3450#0) = 0	AUP(No.3450#0) = 1			
		AUX (No.3405#0) = 0		AUX (No.3405#0) = 1	
		Metric	Inch	Metric	Inch
IS-A	0	2		2	3
IS-B		3		3	4
IS-C		4		4	5
IS-D		5		5	6
IS-E		6		6	7

NOTE
5 When bit 1 (FR3) of parameter No. 1405 is 1, the values in the table need to be incremented by 1.
6 When calculator-type decimal notation is used (bit 0 (DPI) of parameter No. 3401 is set to 1), the number of decimal places is 0.

T

When a value is specified with no decimal point, the number of decimal places is determined as follows.

Address	For a non-axis address	For an axis address
H, M, Q, S, or T	0	
D	α	
R	0 or α (NOTE 6)	
A, B, C, I, J, K, U, V, W, X, Y, or Z	α (NOTE 1)	β (NOTE 2)
Second auxiliary function	γ (NOTE 3)	

Address	Metric input	Inch input
E, F (G98 mode)	0 (NOTE 4)	2 (NOTE 4)
E, F (G99 mode)	4	6

NOTE
1 α is determined according to the increment system for the reference axis (axis specified with parameter No. 1031) as listed in the table in NOTE 2.
2 β is determined according to the increment system for the corresponding axis address as listed in the following table.

Increment system	Linear axis (metric input)	Linear axis (inch input)	Rotation axis
IS-A	2	3	2
IS-B	3	4	3
IS-C	4	5	4
IS-D	5	6	5
IS-E	6	7	6

When bit 7 (IPR) of parameter No. 1004 is set to 1, the above values from which 1 is subtracted are used. When the increment system for an axis is IS-A, however, the setting of bit 7 (IPR) of parameter No. 1004 is not effective.
When calculator-type decimal notation for each axis is used (bit 0 (ADX) of parameter No. 3455 is set to 1), the number of decimal places is 0. When bit 7 (EAP) of parameter No. 3452 is set to 1, however, calculator-type decimal notation is not effective and the number of decimal places is determined as listed in the above table.
3 γ is determined according to the increment system for the reference axis (axis specified with parameter No. 1031) as listed in the following table. (When bit 7 (BDX) of parameter No. 3450 is set to 1, γ is also determined in the same way.)

Increment system for the reference axis	AUP (No.3450#0) = 0	AUP(No.3450#0) = 1			
		AUX (No.3405#0) = 0		AUX (No.3405#0) = 1	
		Metric	Inch	Metric	Inch
IS-A	0	2		2	3
IS-B		3		3	4
IS-C		4		4	5
IS-D		5		5	6
IS-E		6		6	7

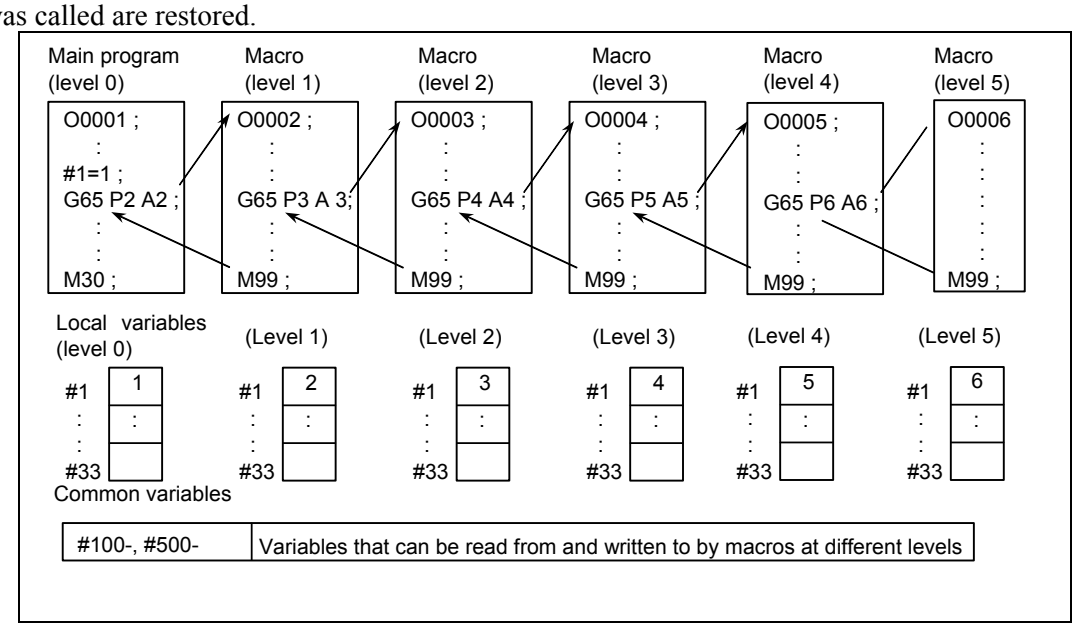
NOTE
4 When bit 2 (FM3) of parameter No. 1404 is 1, the values in the table need to be incremented by 3.
5 When calculator-type decimal notation is used (bit 0 (DPI) of parameter No. 3401 is set to 1), the number of decimal places is 0.
6 If bit 2 (DPD) of parameter No. 6019 is 0, the number of decimal places is 0, and if DPD is 1, it is α.

- Call nesting

Macro calls can be nested to a depth of up to five levels including simple calls (G65) and modal calls (G66/G66.1). Subprogram calls can be nested to a depth of up to 15 levels including macro calls.
A macro program can also be called during MDI operation in the same way.

- Local variable levels

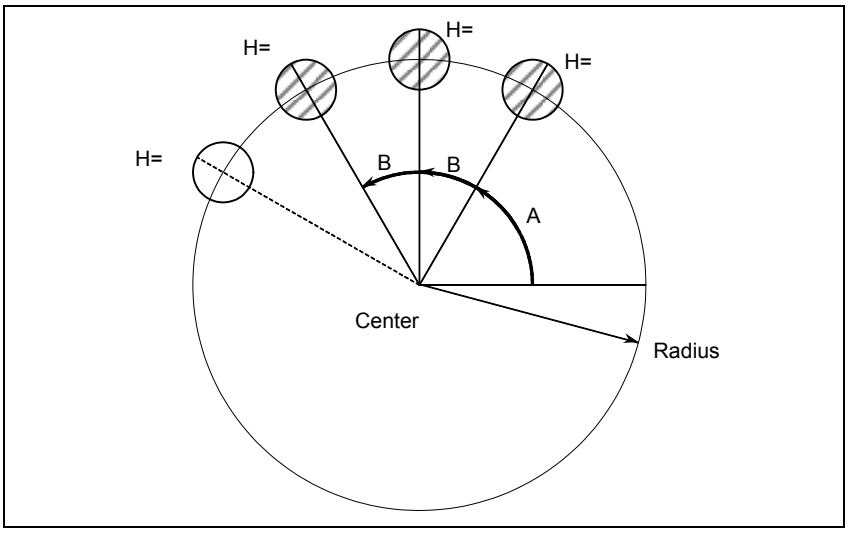
- Local variables from level 0 to 5 are provided for nesting.
- The level of the main program is 0.
- Each time a macro is called (with G66, G66.1, Ggg, or Mmm), the local variable level is incremented by one. The values of the local variables at the previous level are saved in the CNC.
- When M99 is executed in a macro program, control returns to the calling program. At that time, the local variable level is decremented by one; the values of the local variables saved when the macro was called are restored.



Sample program (bolt hole circle)

M

A macro is created which drills H holes at intervals of B degrees after a start angle of A degrees along the periphery of a circle with radius I.
The center of the circle is (X,Y). Commands can be specified in either the absolute or incremental programming.
To drill in the clockwise direction, specify a negative value for B.



- Calling format

```
G65 P9100 Xx Yy Zz Rr Ff Ii Aa Bb Hh ;
```

X : X coordinate of the center of the circle (absolute or incremental programming) ..... (#24)
Y : Y coordinate of the center of the circle (absolute or incremental programming) ..... (#25)
Z : Hole depth ..... (#26)
R : Coordinates of an approach point ..... (#18)
F : Cutting feedrate ..... (#9)
I : Radius of the circle ..... (#4)
A : Drilling start angle ..... (#1)
B : Incremental angle (clockwise when a negative value is specified) ..... (#2)
H : Number of holes ..... (#11)

- Program calling a macro program

```
O0002;
G90 G92 X0 Y0 Z100.0;
G65 P9100 X100.0 Y50.0 R30.0 Z-50.0 F500 I100.0 A0 B45.0 H5;
M30;
```

- Macro program (called program)

```
O9100;
#3=#4003; ..... Stores G code of group 3.
G81 Z#26 R#18 F#9 K0; (Note) .. Drilling cycle.
                                Note: L0 can also be used.
IF[#3 EQ 90]GOTO 1; ..... Branches to N1 in the G90 mode.
#24=#5001+#24; ..... Calculates the X coordinate of the center.
#25=#5002+#25; ..... Calculates the Y coordinate of the center.
N1 WHILE[#11 GT 0]DO 1; ..... Until the number of remaining holes reaches 0
#5=#24+#4*COS[#1]; ..... Calculates a drilling position on the X-axis.
#6=#25+#4*SIN[#1]; ..... Calculates a drilling position on the Y-axis.
G90 X#5 Y#6; ..... Performs drilling after moving to the target position.
#1=#1+#2; ..... Updates the angle.
#11=#11-1; ..... Decrements the number of holes.
END 1;
G#3 G80; ..... Returns the G code to the original state.
M99;
```

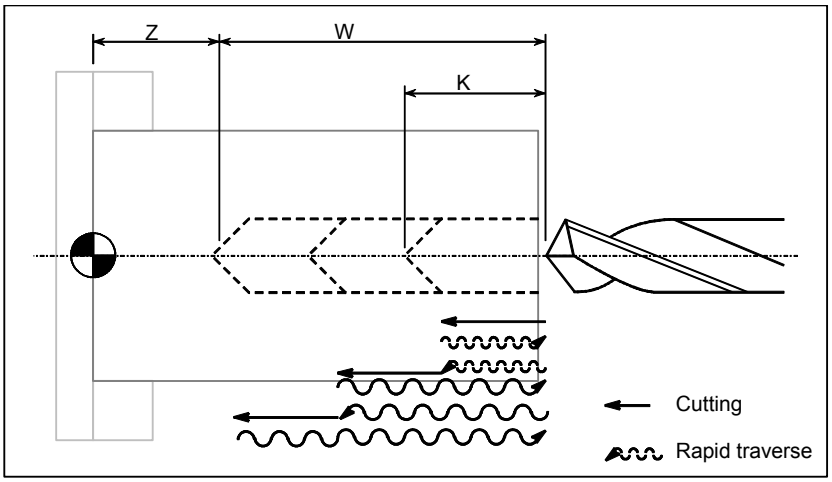
Meaning of variables:

#3: Stores the G code of group 3.
#5: X coordinate of the next hole to drill
#6: Y coordinate of the next hole to drill

Sample program (Drill cycle)

T

Move the tool beforehand along the X- and Z-axes to the position where a drilling cycle starts. Specify Z or W for the depth of a hole, K for the depth of a cut, and F for the cutting feedrate to drill the hole.



- Calling format

G65 P9100 { Zz / Ww } Kk Ff ;

Z : Hole depth (absolute programming)
W : Hole depth (incremental programming)
K : Cutting amount per cycle
F : Cutting feedrate

- Program calling a macro program

```
O0002;
G50 X100.0 Z200.0 ;
G00 X0 Z102.0 S1000 M03 ;
G65 P9100 Z50.0 K20.0 F0.3 ;
G00 X100.0 Z200.0 M05 ;
M30
```

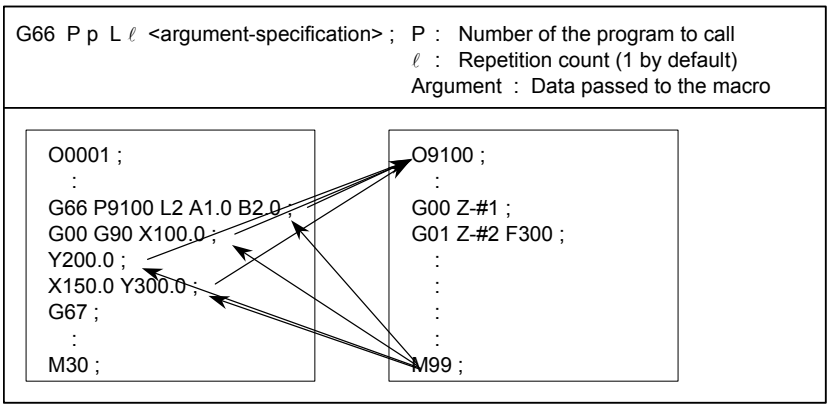
- Macro program (called program)

```
   O9100;
   #1=0 ;..... Clear the data for the depth of the current hole.
   #2=0 ;..... Clear the data for the depth of the preceding hole.
   IF [#23 NE #0] GOTO 1 ;..... If incremental programming, specifies the jump to
                                N1.
   IF [#26 EQ #0] GOTO 8 ; ..... If neither Z nor W is specified, an error occurs.
   #23=#5002-#26 ; ..... Calculates the depth of a hole.
N1 #1=#1+#6 ;..... Calculates the depth of the current hole.
   IF [#1 LE #23] GOTO 2 ; ..... Determines whether the hole to be cut is too deep?
   #1=#23 ;..... Clamps at the depth of the current hole.
N2 G00 W-#2 ; ..... Moves the tool to the depth of the preceding hole at
                    the cutting feedrate.
```

G01 W- [#1-#2] F#9 ;..... Drills the hole.
G00 W#1 ;..... Moves the tool to the drilling start point.
IF [#1 GE #23] GOTO 9 ; ..... Checks whether drilling is completed.
#2=#1 ;..... Stores the depth of the current hole.
GOTO 1 ;
N9 M99
N8 #3000=1 (NOT Z OR W COMMAND).. Issues an alarm.

16.9.2 Modal Call: Call After the Move Command (G66)

Once G66 is issued to specify a modal call a macro is called after a block specifying movement along axes is executed. This continues until G67 is issued to cancel a modal call.



Explanation

- Call

- After G66, specify at address P a program number subject to a modal call.
- When a number of repetitions is required, a number from 1 to 999999999 can be specified at address L.
- As with a simple call (G65), data passed to a macro program is specified in arguments.
- In the G66 mode, a macro or subprogram can be called.

- Cancellation

When a G67 code is specified, modal macro calls are no longer performed in subsequent blocks.

- Call nesting

Macro calls can be nested to a depth of up to five levels including simple calls (G65) and modal calls (G66/G66.1). Subprogram calls can be nested to a depth of up to 15 levels including macro calls.

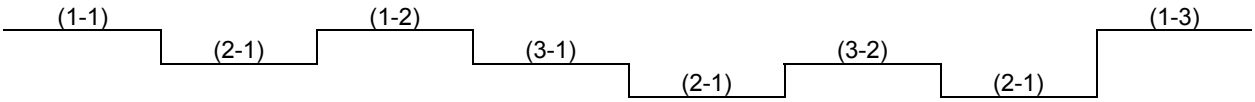
- Modal call nesting

For a single modal call (when G66 is specified only once), each time the move command is executed, the specified macro is called. When nested modal macro calls are specified, the macro at the next higher level is called each time the move command for a macro call is executed.
Macros are called in reverse order in which they are specified. Each time G67 is issued, the macros are canceled one by one in reverse order in which they are specified.
[Example]

```
G66 P9100 ;                O9100 ;                O9200 ;
X10.0 ;      (1-1)         Z50.0 ;      (2-1)     X60.0 ;      (3-1)
G66 P9200 ;                M99 ;                  Y70.0 ;      (3-2)
X15.0 ;      (1-2)                                M99;
G67 ;        Cancels P9200.
G67 ;        Cancels P9100.
```

X-25.0 ; (1-3)

Execution order of the above program (blocks not containing the move command omitted)



* No modal call is performed after (1-3) because the mode is not the macro call mode.

Limitation

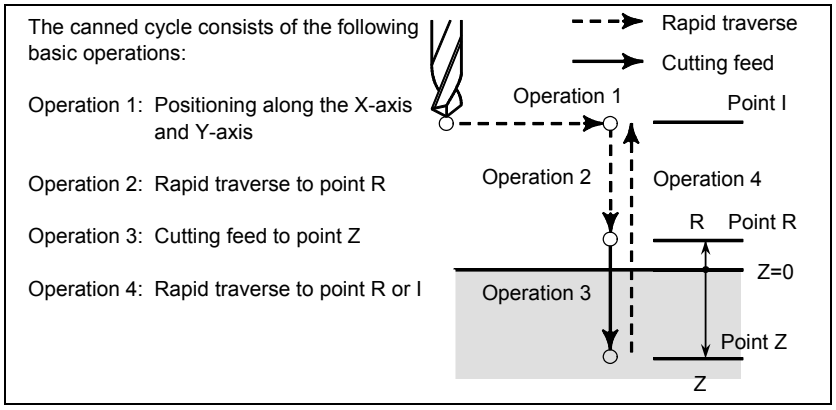
- G66 and G67 blocks are specified in pairs in the same program. If a G67 code is specified not in the G66 mode, an alarm PS1100, "CANCEL WITHOUT MODAL CALL" occurs. Bit 0 (G67) of parameter No. 6000 can be set to 1 to specify that the alarm does not occur in this case.
- In a G66 block, no macros can be called. Local variables (arguments) are set, however.
- G66 needs to be specified before any arguments.
- No macros can be called in a block which contains a code such as a auxiliary function that does not involve movement along an axis.
- Local variables (arguments) can only be set in G66 blocks. Note that local variables are not set each time a modal call is performed.

NOTE
If M99 is specified in a block in which a call is performed, it is executed after the call is performed.

Sample program

M

The same operation as the drilling canned cycle G81 is created using a custom macro and the machining program makes a modal macro call. For program simplicity, all drilling data is specified using absolute values.



- Calling format

```
G66 P9110 Zz Rr Ff Ll ;
```

Z : Coordinates of position Z (absolute programming only).....(#26)
R : Coordinates of position R (absolute programming only).....(#18)
F : Cutting feedrate.....(#9)
L : Repetition count

- Program that calls a macro program

O0001;

```
G28 G91 X0 Y0 Z0;
G92 X0 Y0 Z50.0;
G00 G90 X100.0 Y50.0;
G66 P9110 Z-20.0 R5.0 F500;
G90 X20.0 Y20.0;
X50.0;
Y50.0;
X70.0 Y80.0;
G67;
M30;
```

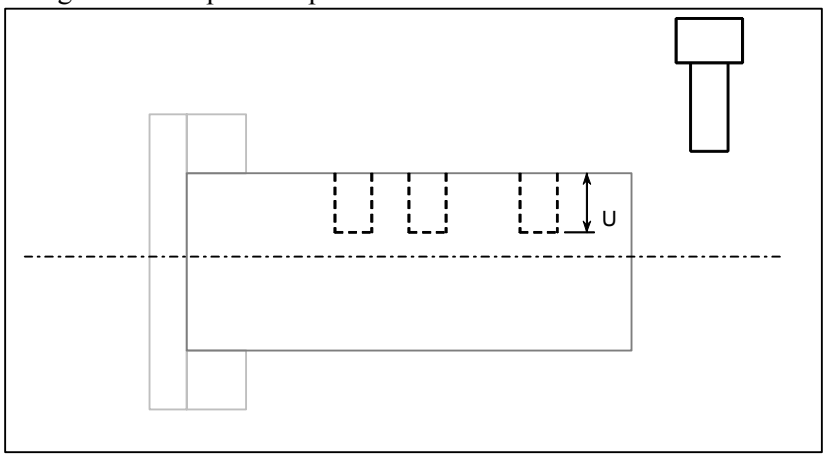
- Macro program (program called)

```
O9110;
    #1=#4001; .....Stores G00/G01.
    #3=#4003;.....Stores G90/G91.
    #4=#4109;.....Stores the cutting feedrate.
    #5=#5003;.....Stores the Z coordinate at the start of drilling.
    G00 G90 Z#18; .....Positioning at position R
    G01 Z#26 F#9;.....Cutting feed to position Z
    IF[#4010 EQ 98]GOTO 1; ...Return to position I
    G00 Z#18;.....Positioning at position R
    GOTO 2;
N1 G00 Z#5;.....Positioning at position I
N2 G#1 G#3 F#4;.....Restores modal information.
    M99;
```

Sample program

T

This program makes a groove at a specified position.



- Calling format

```
G66 P9110 Uu Ff
```

U : Groove depth (incremental programming)
F : Cutting feed of grooving

- Program that calls a macro program

```
O0003 ;
G50 X100.0 Z200.0 ;
S1000 M03 ;
```

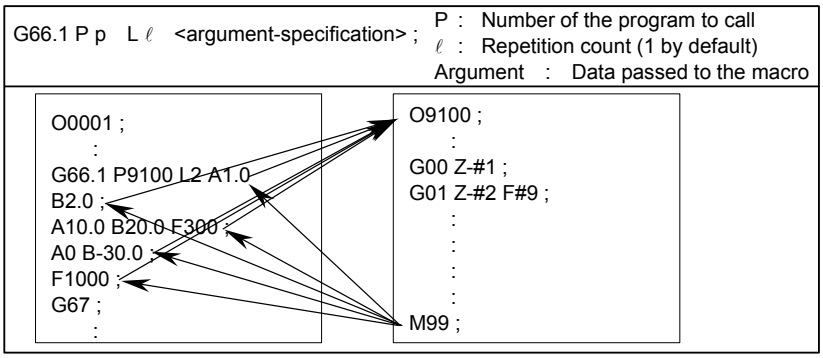
```
G66 P9110 U5.0 F0.5 ;
G00 X60.0 Z80.0 ;
Z50.0 ;
Z30.0 ;
G67 ;
G00 X00.0 Z200.0 M05 ;
M30;
```

- Macro program (program called)

```
O9110 ;
G01 U - #21 F#9 ; ..... Cuts the workpiece.
G00 U#21 ;..... Retracts the tool.
M99 ;
```

16.9.3 Modal Call: Each Block Call (G66.1)

In this macro call mode, the specified macro is unconditionally called for each NC command block. All data other than O, file name, N, and G codes that is specified in each block is not executed and is used as arguments. (The G code in the block in which G66.1 is specified is not used as an argument. Only the last G code specified in subsequent blocks is used as an argument.)
The NC command blocks having O or file name N have the same effect as when G65P is specified in the next position, the other NC command blocks have the same effect as when G65P is specified at the beginning.



[Example]

In the G66.1 P100 ; mode,
N001 G01 G91 X100 Y200 D1 R1000 ;
is the same as
N001 G65 P100 G01 G91 X100 Y200 D1 R1000 ;

Explanation

- Call

- After G66.1, specify at address P a program number subject to a modal call.
- When a number of repetitions is required, a number from 1 to 999999999 can be specified at address L.
- As with a simple call (G65), data passed to a macro program is specified in arguments.
- In the G66.1 mode, a macro or subprogram can be called.

- Cancellation

When a G67 code is specified, modal macro calls are no longer performed in subsequent blocks.

- Call nesting

Macro calls can be nested to a depth of up to five levels including simple calls (G65) and modal calls (G66/G66.1). Subprogram calls can be nested to a depth of up to 15 levels including macro calls.

- Modal call nesting

For a single modal call (when G66.1 is specified only once), the specified macro is called for each NC command block. When nested modal macro calls are specified, the macro at the next higher level is also called in a block within a called macro in which an NC command is specified.
Macros are called in reverse order in which they are specified. Each time G67 is issued, the macros are canceled one by one in reverse order in which they are specified.

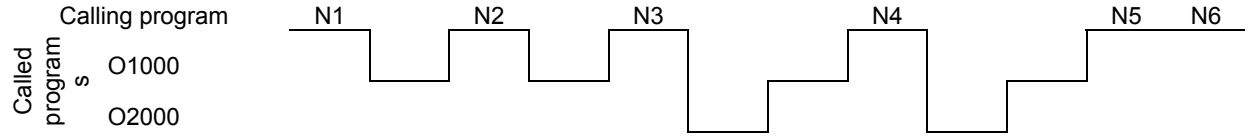
[Example] Axis specification switching

N1 G66.1 P1000 X10.0 ;	→	Calls O1000 and executes Y10.0.
N2 X20.0 ;	→	Calls O1000 and executes Y20.0.
N3 G66.1 P2000 Y10.0 Z20.0 ;	→	Calls O2000 and executes Y20.0 Z10.0. Then, calls O1000 and executes X20.0 Z10.0.
N4 X10.0 Y20.0 Z30.0 ;	→	Calls O2000 and executes X10.0 Y30.0 Z20.0. Then, calls O1000 and executes X30.0 Y10.0 Z20.0.
N5 G67 ;	→	Cancels P2000.
N6 G67 ;	→	Cancels P1000.

```
O1000 X#25 Y#24 Z#26 ;          (X-Y switching)
M99 ;

O2000 X#24 Y#26 Z#25 ;          (Y-Z switching)
M99 ;
```

Execution order of the above program (blocks not containing the move command omitted)



In N1 and N2 blocks, O1000 is called and the X and Y specifications are made to change places.
In N3 and N4 blocks, O2000 is called first and the Y and Z specifications are made to change places. For the switched specification, O1000 is called and the X and Y specifications are made to change places. Therefore, the X, Y, and Z specifications are switched to the Z, X, and Y specifications.

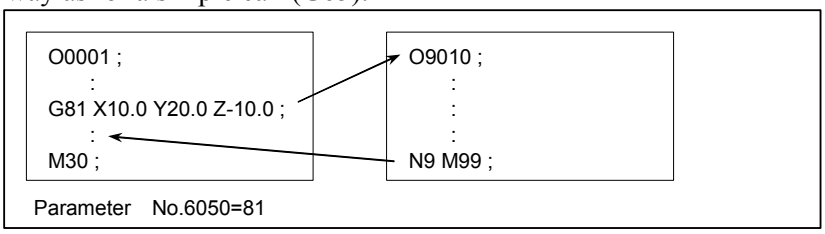
Limitation

- G66.1 and G67 blocks are specified in pairs in the same program. If a G67 code is specified not in the G66.1 mode, an alarm PS1100, “CANCEL WITHOUT MODAL CALL” occurs. Bit 0 (G67) of parameter No. 6000 can be set to 1 to specify that the alarm does not occur in this case.
- G66.1 block
  (a) In a G66.1 block, a macro is called.
  (b) The correspondence between addresses specified as arguments and variables is the same as for simple calls.
- Block following a G66.1 block in which a call is performed (not including G66.1 blocks)
  (a) Addresses G, P, and L can also be used as arguments. Address G corresponds to #10; address L to #12; address P to #16. However, the restrictions on the input format of normal NC commands are put on the data. For example, ;G1000. P0.12 L-4 cannot be specified.
  (b) When multiple G codes are specified, only the last G code is used as an argument. O, file name, and N codes and G codes not in group 00 are passed to the next and subsequent blocks.

NOTE
1 In a block in which only an O number, file name, sequence number, EOB, macro statement, or M99 command is specified, a macro is not called for each block.
2 In each block, when an address other than O, file name, or N is specified, it is assumed to be an NC command and a macro is called for each block. When N is specified following an address other than O, file name, or N, it is used as an argument. In this case, N corresponds to variable #14 and the number of decimal places is 0.
3 If M99 is specified in a block in which a macro is called, it is executed after the call is performed.

16.9.4 Macro Call Using a G Code

By setting a G code number used to call a macro program in a parameter, the macro program can be called in the same way as for a simple call (G65).



Explanation

By setting a G code number from -9999 to 9999 used to call a custom macro program (O9010 to O9019) in the corresponding parameters Nos.6050 to 6059, the macro program can be called in the same way as with G65. To call custom macro program O9040 to O9049 using a G code with a decimal point, set bit 0 (DPG) of parameter No. 6007 to 1 and set the G code number in the corresponding parameters Nos. 6060 to 6069.

The number of decimal places of a G code is 1. Set the value obtained by multiplying a desired G code number by 10 in the corresponding parameter.

[Example] When parameter No. 6060 is set to 234, custom macro program O9040 is called using G23.4.

When a negative G code is set, a modal call is performed. In this case, bit 3 (MGE) of parameter No. 6007 can be set to select the G66 or G66.1 mode.

For example, when a parameter is set so that macro program O9010 can be called with G81, a user-specific cycle created using a custom macro can be called without modifying the machining program.

- Correspondence between parameter numbers and program numbers

G code with no decimal point		G code with a decimal point	
Parameter number	Program number	Parameter number	Program number
6050	O9010	6060	O9040
6051	O9011	6061	O9041
6052	O9012	6062	O9042
6053	O9013	6063	O9043
6054	O9014	6064	O9044
6055	O9015	6065	O9045
6056	O9016	6066	O9046
6057	O9017	6067	O9047
6058	O9018	6068	O9048
6059	O9019	6069	O9049

- Repetition

As with a simple call, a number of repetitions from 1 to 999999999 can be specified at address L.

- Argument specification

As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used.

Limitation

- Nesting of calls using G codes

- To call another program in a program called using a G code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using an M, T, or S code, second auxiliary function, or specific code can be performed in a program called using a G code.

16.9.5 Macro Call Using a G Code (Specification of Multiple Definitions)

By setting the starting G code number used to call a macro program, the number of the starting program to be called, and the number of definitions, macro calls using multiple G codes can be defined.

Explanation

As many custom macros as the number specified in parameter No. 6040 can be called using as many G codes as the number specified in parameter No. 6040. The numeric value set in parameter No. 6038 indicates the starting G code number and the program number set in parameter No. 6039 indicates the starting program number. To disable this type of call, set 0 in parameter No. 6040.

When a negative G code is set in parameter No. 6038, modal calls are performed. In this case, bit 3 (MGE) of parameter No. 6007 can be set to select the G66 or G66.1 mode.

The number of repetitions and argument specification are set in the same way as with a macro call using a G code.

[Example]

Set parameter No. 6038 to 900, parameter No. 6039 to 10000000, and parameter No. 6040 to 100.

G900 → O1000
G901 → O1001
G902 → O1002
:
G999 → O1099

Custom macro calls (simple calls) for 100 combinations are defined as shown above. When parameter No. 6038 is changed to -900, custom macro calls (modal calls) for the same combinations are defined.

NOTE

1 The calls defined by this setting become all invalid in the following cases:
<1> A value outside the valid data range is set in one of the above parameters.
<2> (No. 6039 + No. 6040 - 1) > 99999999
2 Simple and modal calls cannot be mixed in the specification.
3 If the G code set in parameters Nos. 6050 to 6059 to call the corresponding macro program is within the G code range for calling programs using multiple G codes, the macro program corresponding to the G code set in parameters Nos. 6050 to 6059 is called.

16.9.6 Macro Call Using a G Code with a Decimal Point (Specification of Multiple Definitions)

When bit 0 (DPG) of parameter No. 6007, by setting the starting G code number with a decimal point used to call a macro program, the number of the starting program to be called, and the number of definitions, multiple macro calls using multiple G codes with a decimal point can be defined.

Explanation

As many custom macros as the number specified in parameter No. 6043 can be called using as many G codes with a decimal point as the number specified in parameter No. 6043. The numeric value set in parameter No. 6041 indicates the starting G code number with a decimal point and the program number set in parameter No. 6042 indicates the starting program number. To disable this type of call, set 0 in parameter No. 6043.

When a negative G code is set in parameter No. 6041, modal calls are performed. In this case, bit 3 (MGE) of parameter No. 6007 can be set to select the G66 or G66.1 mode.

The number of repetitions and argument specification are set in the same way as with a macro call using a G code.

[Example]

Set parameter No. 6041 to 900, parameter No. 6042 to 2000, and parameter No. 6043 to 100.

G90.0 → O2000
G90.1 → O2001
G90.2 → O2002
:
G99.9 → O2099

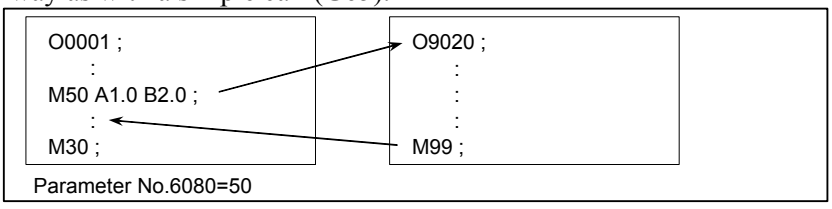
Custom macro calls (simple calls) for 100 combinations are defined as shown above. When parameter No. 6041 is changed to -900, custom macro calls (modal calls) for the same combinations are defined.

NOTE

1 The calls defined by this setting become all invalid in the following cases:
  <1> A value outside the valid data range is set in one of the above parameters.
  <2> (No. 6042 + No. 6043 - 1) > 99999999
  <3> Bit 0 (DPG) of parameter No. 6007 is set to 0 (this setting invalidates a macro call using a G code with a decimal point).
2 Simple and modal calls cannot be mixed in the specification.
3 If the G code set in parameters Nos. 6060 to 6069 to call the corresponding macro program is within the G code range for calling programs using multiple G codes, the macro program corresponding to the G code set in parameters Nos. 6060 to 6069 is called.

16.9.7 Macro Call Using an M Code

By setting an M code number used to call a macro program in a parameter, the macro program can be called in the same way as with a simple call (G65).



Explanation

By setting an M code number from 3 to 99999999 used to call custom macro program O9020 to O9029 in the corresponding parameters Nos. 6080 to 6089, the macro program can be called in the same way as with G65.

- Correspondence between parameter numbers and program numbers

Parameter number	Corresponding program number
6080	O9020
6081	O9021
6082	O9022
6083	O9023
6084	O9024
6085	O9025
6086	O9026
6087	O9027
6088	O9028
6089	O9029

Example) When parameter No. 6080 is set to 990, O9020 is called using M990.

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used. When the parameter MAA (No.6009#2) is set to 1, address G becomes an argument, too.

1. When parameter MAA (No.6009#2) is set to 0
   The address commanded after call code become arguments. Argument specification I or argument specification II are used.
   Argument specification I

Address	Variable number	Address	Variable number	Address	Variable number
A	#1	J	#5	S	#19
B	#2	K	#6	T	#20
C	#3	L	*2	U	#21
D	#7	M	#13 *3	V	#22
E	#8	M(Call code)	*4	W	#23
F	#9	N	#14 *5	X	#24
G	*1	P	#16	Y	#25
H	#11	Q	#17	Z	#26
I	#4	R	#18		

   *1: Address G can not command. If address G is commanded, alarm PS0129 occurs.
   *2: Address L is a number of times in which the macro call is repeated.
   *3: Address M other than call code is passed to variable #13.
   *4: Address M for call code does not become an argument.
   *5: Address N is passed to variable #14.

2. When parameter MAA (No.6009#2) is set to 1
   The address commanded after call code become arguments. Address G becomes an argument, too. Argument specification I or argument specification II are used.
   Argument specification I

Address	Variable number
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
G	#28 to #32 *1
H	#11
I	#4

Address	Variable number
J	#5
K	#6
L	*2
M	#13 *3
M(Call code)	*4
N	#14 *5
P	#16
Q	#17
R	#18

Address	Variable number
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

*1: The first five addresses G in ascending order of G code groups are used as arguments and passed to variables #28 to #32.
*2: Address L is a number of times in which the macro call is repeated.
*3: Address M other than call code is passed to variable #13.
*4: Address M for call code does not become an argument.
*5: Address N is passed to variable #14.

Limitation

- An M code used to call a macro program must be specified at the start of a block.
- To call another program in a program called using an M code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using an M code.

16.9.8 Macro Call Using an M Code (Specification of Multiple Definitions)

By setting the starting M code number used to call a macro program, the number of the starting program to be called, and the number of definitions, macro calls using multiple M codes can be defined.

Explanation

As many custom macros as the number specified in parameter No. 6049 can be called using as many M codes as the number specified in parameter No. 6049. The numeric value set in parameter No. 6047 indicates the starting M code number and the program number set in parameter No. 6048 indicates the starting program number. To disable this type of call, set 0 in parameter No. 6049.
The number of repetitions and argument specification are set in the same way as with a macro call using a M code.
[Example]

Set parameter No. 6047 to 9000, parameter No. 6048 to 4000, and parameter No. 6049 to 100.
M90000000 → O4000
M90000001 → O4001
M90000002 → O4002
:
M90000099 → O4099

Custom macro calls (simple calls) for 100 combinations are defined as shown above.

NOTE
1 The calls defined by this setting become all invalid in the following cases:
<1> A value outside the valid data range is set in one of the above parameters.
<2> (No. 6048 + No. 6049 - 1) > 99999999

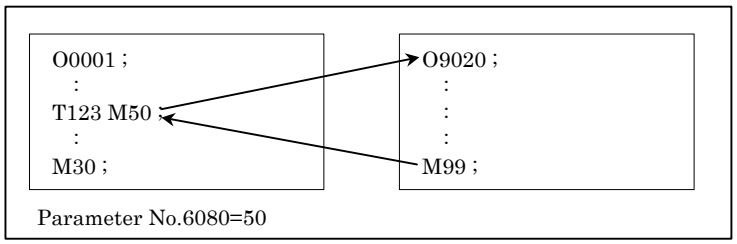
NOTE
2 If the M code set in parameters Nos. 6080 to 6089 to call the corresponding macro program is within the M code range for calling programs using multiple M codes, the macro program corresponding to the M code set in parameters Nos. 6080 to 6089 is called.

16.9.9 Special Macro Call using M code

Usually, it is necessary to command M code for Macro Call using M code or Macro Call using M code (Specification of 3 sets) at the head of the block. Alarm PS0127 occurs if M code is commanded for macro call at the position of the block that is not the head.
Special Macro Call using M code is a function to execute macro call when call code is commanded at the position of the block that is not the head. By setting parameter MSM (No.6009#0) is set to 1, it becomes possible.

Example

When M code is commanded for a macro call at the position of the block that is not the head



When parameter MSM (No.6009#0) is set to 0, PS0127 alarm occurs.
When parameter MSM (No.6009#0) is set to 1, Macro Call using M code is executed. And, Address T becomes an argument and #20 is set to 123.0.

Explanation

When Special Macro Call using M code becomes possible because parameter MSM (No.6009#0) is set to 1, the execution of macro call changes by the position in which M code for macro call is commanded.

1. When M code for macro call is commanded at the head of the block
   It operates as usual Macro Call.
   For details, see Subsection 16.9.7, "Macro Call Using an M Code".
2. When M code for macro call is commanded at the position of the block that is not the head
   It operates as Special Macro Call using M code.

- Repetition

Address L is also used as an argument, so that no repetition count can be specified.

- Argument specification

All addresses are treated as arguments. However, the specifiable addresses and specification range conform to the specification address range of the CNC. Only argument specification I can be used.
When the parameter MAA (No.6009#2) is set to 1, address G becomes an argument, too.

1. When parameter MAA (No.6009#2) is set to 0
   All addresses other than M for call code, G and L are treated as arguments. However, the specifiable addresses and specification range conform to the specification address range of the CNC. Only argument specification I can be used.
   Argument specification I

Address	Variable number	Address	Variable number	Address	Variable number
A	#1	J	#5	S	#19
B	#2	K	#6	T	#20
C	#3	L	*2	U	#21
D	#7	M	#13 *3	V	#22
E	#8	M(Call code)	*4	W	#23
F	#9	N	#14 *5	X	#24
G	*1	P	#16	Y	#25
H	#11	Q	#17	Z	#26
I	#4	R	#18		

*1: Even if address G is commanded, PS alarm does not occur. And address G does not become an argument.
*2: Address L does not become a number of times in which the macro call is repeated, and does not become an argument.
*3: Address M other than call code is passed to variable #13.
*4: Address M for call code does not become an argument.
*5: Address N is passed to variable #14, and becomes a sequence number.

2. When parameter MAA (No.6009#2) is set to 1
All addresses are treated as arguments. However, the specifiable addresses and specification range conform to the specification address range of the CNC. Only argument specification I can be used. Address G, L and M for call code become arguments, too.
Argument specification I

Address	Variable number	Address	Variable number	Address	Variable number
A	#1	J	#5	S	#19
B	#2	K	#6	T	#20
C	#3	L	#12 *2	U	#21
D	#7	M	#13 *3	V	#22
E	#8	M(Call code)	#27 *4	W	#23
F	#9	N	#14 *5	X	#24
G	#28 to #32 *1	P	#16	Y	#25
H	#11	Q	#17	Z	#26
I	#4	R	#18		

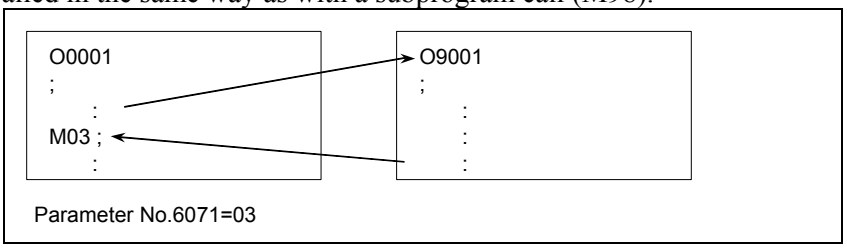
*1: The first five addresses G in ascending order of G code groups are used as arguments and passed to variables #28 to #32.
*2: Address L is passed to variable #12, and does not become a number of times in which the macro call is repeated.
*3: Address M other than call code is passed to variable #13.
*4: Address M for call code is passed to variable #27.
*5: Address N is passed to variable #14, and becomes a sequence number.

Limitation

- To call another program in a program called using an M code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using an M code.
- When multiple call commands are specified in the same block, the first call code is selected.

16.9.10 Subprogram Call Using an M Code

By setting an M code number used to call a subprogram (macro program) in a parameter, the macro program can be called in the same way as with a subprogram call (M98).



Explanation

By setting an M code number from 3 to 99999999 used to call subprogram O9001 to O9009 in the corresponding parameters Nos. 6071 to 6079, the subprogram can be called in the same way as with M98.

- Correspondence between parameter numbers and program numbers

Parameter number	Program number
6071	O9001
6072	O9002
6073	O9003
6074	O9004
6075	O9005
6076	O9006
6077	O9007
6078	O9008
6079	O9009

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

Argument specification is not allowed.

- M code

An M code in a macro program that has been called is treated as an ordinary M code.

Limitation

- To call another program in a program called using an M code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using an M code.

16.9.11 Subprogram Call Using an M Code (Specification of Multiple Definitions)

By setting the starting M code number used to call a subprogram, the number of the starting subprogram to be called, and the number of definitions, subprogram calls using multiple M codes can be defined.

Explanation

As many subprograms as the number specified in parameter No. 6046 can be called using as many M codes as the number specified in parameter No. 6046. The numeric value set in parameter No. 6044 indicates the starting M code number and the numeric value set in parameter No. 6045 indicates the starting subprogram number. To disable this type of call, set 0 in parameter No. 6046.

[Example]

Set parameter No. 6044 to 80000000, parameter No. 6045 to 30000000, and parameter No. 6046 to 100.

M80000000 → O3000
M80000001 → O3001
M80000002 → O3002
:
M80000099 → O3099

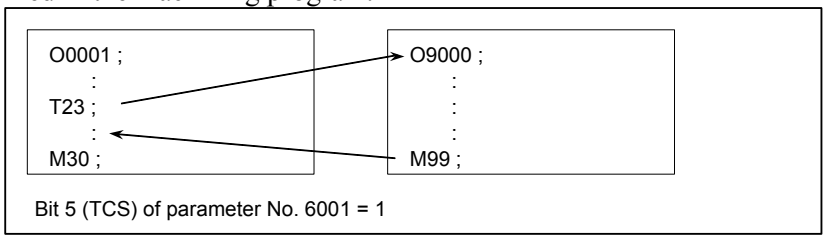
Subprogram calls for 100 combinations are defined as shown above.

NOTE

1 The calls defined by this setting become all invalid in the following cases:
<1> A value outside the valid data range is set in one of the above parameters.
<2> (No. 6045 + No. 6046 - 1) > 99999999
2 If the M code set in parameters Nos. 6071 to 6079 to call the corresponding subprogram is within the M code range for calling subprograms using multiple M codes, the subprogram corresponding to the M code set in parameters Nos. 6071 to 6079 is called.

16.9.12 Subprogram Calls Using a T Code

By enabling subprograms to be called with a T code in a parameter, a subprogram can be called each time the T code is specified in the machining program.



Explanation

- Call

By setting bit 5 (TCS) of parameter No. 6001 to 1, subprogram O9000 can be called each time a T code is specified in a machining program. A T code specified in a machining program is assigned to common variable #149.

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

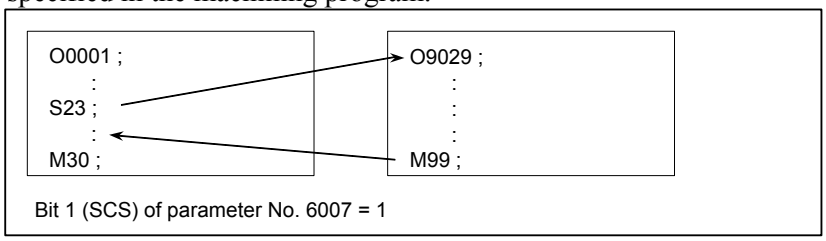
Argument specification is not allowed.

Limitation

- To call another program in a program called using a T code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using a T code.

16.9.13 Subprogram Calls Using an S Code

By enabling subprograms to be called with an S code in a parameter, a subprogram can be called each time the S code is specified in the machining program.



Explanation

- Call

By setting bit 1 (SCS) of parameter No. 6007 to 1, subprogram O9029 can be called each time a S code is specified in a machining program. An S code specified in a machining program is assigned to common variable #147.

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

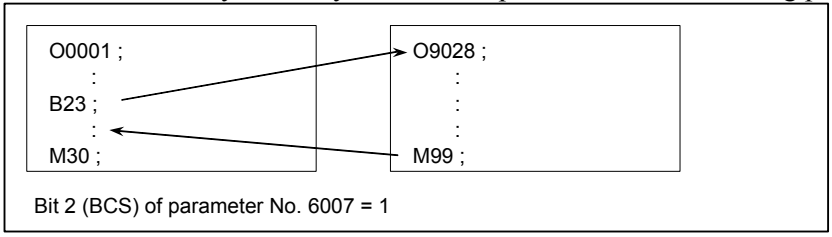
Argument specification is not allowed.

Limitation

- To call another program in a program called using an S code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using an S code.

16.9.14 Subprogram Calls Using a Secondary Auxiliary Function

By enabling subprograms to be called with a secondary auxiliary function in a parameter, a subprogram can be called each time the secondary auxiliary function is specified in the machining program.



Explanation

- Call

By setting bit 2 (BCS) of parameter No. 6007 to 1, subprogram O9028 can be called each time a secondary auxiliary function code is specified in a machining program. A secondary auxiliary function specified in a machining program is assigned to common variable #146.

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

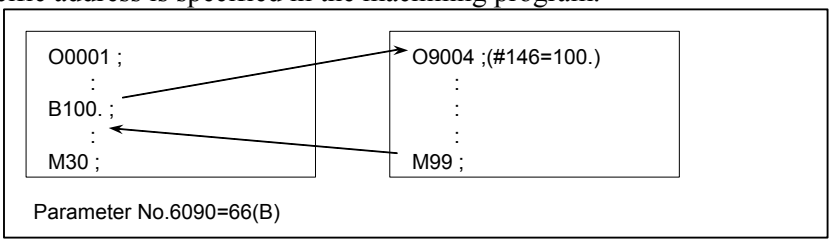
Argument specification is not allowed.

Limitation

- To call another program in a program called using a second auxiliary function, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using a second auxiliary function.

16.9.15 Subprogram Call Using a Specific Address

By enabling subprograms to be called with a specific address in a parameter, a subprogram can be called each time the specific address is specified in the machining program.



Explanation

- Call

By setting the code (ASCII code converted to decimal) corresponding to a specific address in parameter No. 6090 or No. 6091, the custom macro program, O9004 or O9005, corresponding to each parameter can be called when the specific address is specified in a machining program. The code value corresponding to a specific address specified in a machining program is assigned to the common variables (#146, #147). The table below indicates the addresses that can be set.

M

Address	Parameter setting	Address	Parameter setting
A	65	P	80
B	66	Q	81
D	68	R	82
F	70	S	83
H	72	T	84
I	73	V	86
J	74	X	88
K	75	Y	89
L	76	Z	90
M	77		

NOTE
When address L is set, the number of repetitions cannot be set.

T

Address	Parameter setting	Address	Parameter setting
A	65	L	76
B	66	M	77
F	70	P	80
H	72	Q	81
I	73	R	82
J	74	S	83
K	75	T	84

NOTE
When address L is set, the number of repetitions cannot be set.

- Correspondence between parameter numbers and program numbers and between the parameter numbers and common variables

Parameter number	Program number	Common variable
6090	O9004	#146
6091	O9005	#147

- Repetition

As with a simple call, a number of repetitions from 1 to 99999999 can be specified at address L.

- Argument specification

Argument specification is not allowed.

Limitation

- To call another program in a program called using a specific code, only G65, M98, G66, or G66.1 can be used normally.
- When bit 6 (GMP) of parameter No. 6008 is set to 1, a call using a G code can be performed in a program called using a specific code.

Sample program

By using the subprogram call function that uses M codes, the cumulative usage time of each tool is measured.

Conditions

- The cumulative usage time of each of tools T01 to T05 is measured.
  No measurement is made for tools with numbers greater than T05.
- The following variables are used to store the tool numbers and measured times:

#501	Cumulative usage time of tool number 1
#502	Cumulative usage time of tool number 2
#503	Cumulative usage time of tool number 3
#504	Cumulative usage time of tool number 4
#505	Cumulative usage time of tool number 5

- Usage time starts being counted when the M03 command is specified and stops when M05 is specified. System variable #3002 is used to measure the time during which the cycle start lamp is on. The time during which the machine is stopped by feed hold and single block stop operation is not counted, but the time used to change tools and pallets is included.

Operation check

- Parameter setting

Set 3 in parameter No.6071, and set 5 in parameter No.6072.

- Variable value setting

Set 0 in variables #501 to #505.

- Program that calls a macro program

```
O0001;
T01 M06;
M03;
     :
M05;..... Changes #501.
T02 M06;
M03;
     :
M05;..... Changes #502.
T03 M06;
M03;
     :
M05;..... Changes #503.
T04 M06;
M03;
     :
M05;..... Changes #504.
T05 M06;
M03;
     :
M05;..... Changes #505.
M30;
```

- Macro program (program called)

```
O9001(M03);      ..... Macro to start counting
   M01;
   IF[#4120 EQ 0]GOTO 9; ..... No tool specified
   IF[#4120 GT 5]GOTO 9; ..... Out-of-range tool number
   #3002=0;      ..... Clears the timer.
N9 M03;          ..... Rotates the spindle in the forward direction.
   M99;

O9002(M05);      ..... Macro to end counting
   M01;
   IF[#4120 EQ 0]GOTO 9; ..... No tool specified
   IF[#4120 GT 5]GOTO 9; ..... Out-of-range tool number
   #[500+#4120]=#3002+#[500+#4120]; .. Calculates cumulative time.

N9 M05; ..... Stops the spindle.
   M99;
```

16.10 MACRO CALL ARGUMENT FOR AXIS NAME EXPANSION

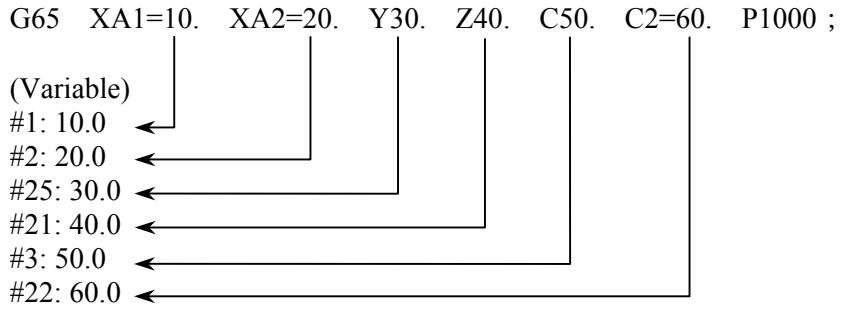
Macro argument can be specified to the address of axis name expansion. By setting the parameter (No.11647), the address of axis name expansion is allocated to local variable number(#1 - #33).
This function is effective to not only an axis name expansion but also a usual axis address of one character. A usual axis address of one character can allocate to the local variable number (#1-#33).

Example

In case of the following axis configuration, if all arguments are specified, the relationship between the parameter (No.11647) and local variable number is as follows.

Axis name	Parameter (No.11647)	Local variable
XA1 (*1)	1	Argument is set to #1
XA2 (*1)	2	Argument is set to #2
Y	0	Argument is set to #25
Z	21	Argument is set to #21
C	0	Argument is set to #3
C2 (*1)	22	Argument is set to #22

(*1) Axis name expansion



Allocating the same local variable

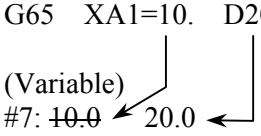
Please do not allocate the same local variable to 2 or more arguments. If the same local variable is allocated, the argument specified later becomes effective.

Example

When 2 arguments of XA1=10. and D20.0 are commanded to local variable #7, the later argument D20.0 becomes effective.

Axis name	Parameter (No.11647)	Local variable
XA1 (*1)	7	Argument is set to #7

(*1) Axis name expansion



Using the same axis name

When using the same axis name, the parameter (No.11647) setting of the smallest axis number becomes effective. The setting of the other axis becomes invalid.

Example

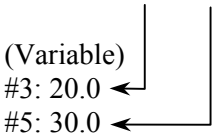
When using the same axis name and using the axis name expansion, the relationship between the parameter (No.11647) and local variable number is as follows.

Axis name	Parameter (No.11647)	Local variable
XA1 (*1)	0	The argument for XA1 is invalid
XA1 (*1)	2	Invalid setting
YA2 (*1)	3	Argument is set to #3
YA2 (*1)	4	Invalid setting
C	5	Argument is set to #5
C	6	Invalid setting

(*1) Axis name expansion

G65 XA1=10. YA2=20. C30. P1000 ;
Because the argument for XA1 is invalid, the alarm (PS0129) occurs.

G65 YA2=20. C30. P1000 ;



The setting range of the parameter

The setting range of the parameter (No.11647) is 0,1-33. This corresponds to local variable number (#1-#33). When other values are set, the setting of the axis is invalid. Therefore, when using the axis name expansion, the alarm (PS0129) occurs by commanding the axis. If the axis does not use the axis name expansion, the argument is assigned to original local variable (#1-#33).

16.11 PROCESSING MACRO STATEMENTS

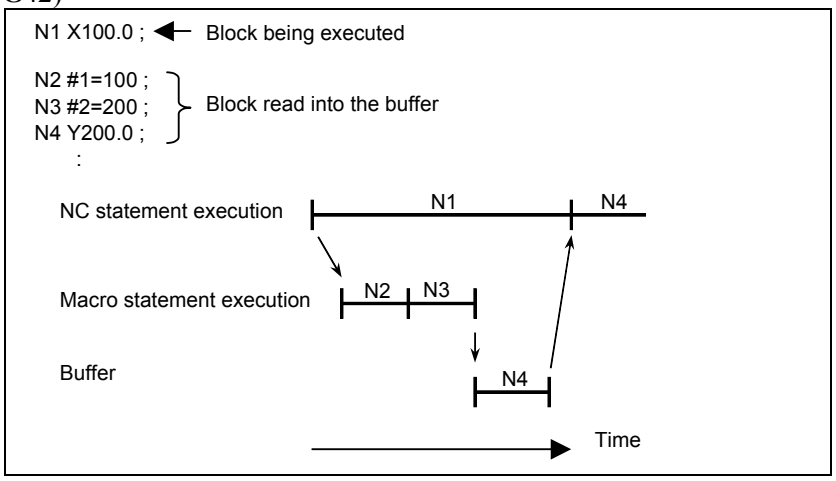
For smooth machining, the CNC prereads the NC statement to be performed next. This operation is referred to as buffering. For example, in prereading due to AI contour control, up to 1000 blocks of NC statements are buffered.
In the cutter compensation mode (G41 or G42), the CNC prereads the NC statements in at least three blocks ahead to calculate intersections.
Macro statements for arithmetic expressions and conditional branches are processed as soon as they are read into the buffer. Therefore, the timing of the macro statement execution is not always the specified order.
At the blocks containing M00, M01, M02 or M30, blocks containing M-codes for which buffering is suppressed by setting parameters Nos. 3411 to 3420, 3421 to 3432, and 11290 to 11299, and blocks containing prevention buffering G codes such as G31 or G53, the CNC stops to preread the NC statement after that. Then, the stop of the macro statement execution is guaranteed until such M codes or G codes complete its execution.

Explanation

- When the next block is buffered

Example 1:

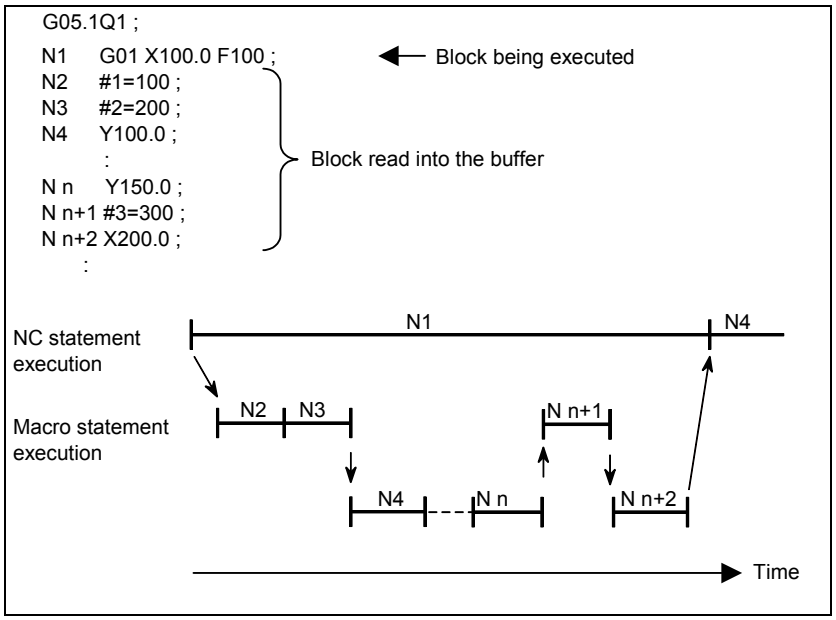
Prereading the next block when the system is not in AI contour control mode or cutter compensation mode (G41, G42)



While N1 is being executed, the next NC statement in N4 is read into the buffer.
The macro statements in N2 and N3 between N1 and N4 are processed while N1 is being executed.

Example 2:

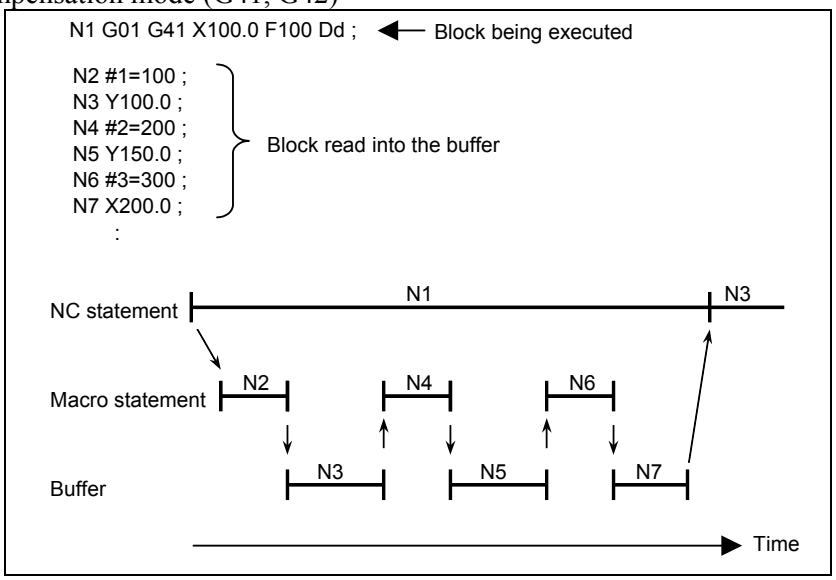
In AI contour control mode



While N1 is being executed, the NC statements in up to the subsequent 1000 blocks (Nn+2) are read into the buffer. The macro statements in N2, N3, and Nn+1 between the blocks are processed while N1 is being executed.

Example 3:

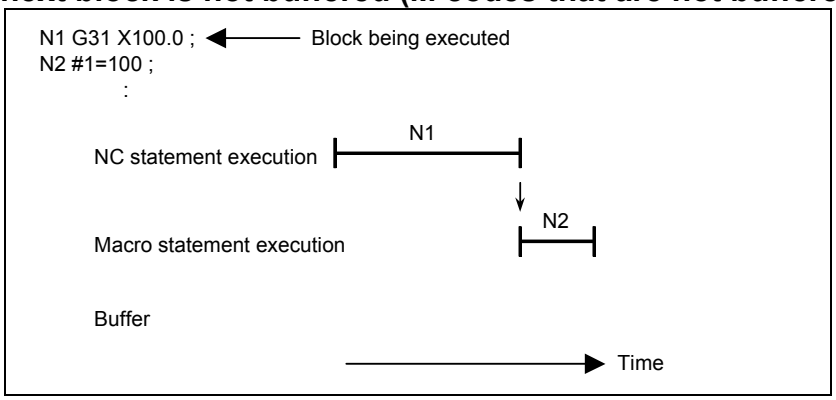
In cutter compensation mode (G41, G42)



While N1 is being executed, the NC statements in the subsequent three blocks (blocks up to N7) are read into the buffer.

The macro statements in N2, N4, and N6 between the blocks are processed while N1 is being executed.

- When the next block is not buffered (M codes that are not buffered, G31, etc.)



⚠CAUTION

In case that you need to execute the macro statement after completing the block just before the macro statement, specify M code or G code that is not buffered just before the macro statement. Specially, in case of reading/writing the system variables to control signals, coordinates, offset value, etc., it may different system variable data by the timing of the NC statement execution. To avoid this phenomenon, specify such M codes or G codes before the macro statement, if necessary.

Diagnosis data

Diagnosis	1493	Number of macro statement blocks executed by custom and execution macros

[Data type] 2-word

[Unit of data] Block

The number of macro statement blocks executed by custom and execution macros per 1024 ms is displayed.

This data provides an indication of the actual macro statement processing speed.

16.12 REGISTERING CUSTOM MACRO PROGRAMS

Custom macro programs are similar to subprograms. They can be registered and edited in the same way as subprograms. The storage capacity is determined by the total length of tape used to store both custom macros and subprograms.

16.13 CODES AND RESERVED WORDS USED IN CUSTOM MACROS

In addition to the codes used in ordinary programs, the following codes are used in custom macro programs.

Explanation

- Codes

(1) When the ISO code is used or when bit 4 (ISO) of parameter No. 6008 is set to 0
(The codes are represented in hexadecimal.)

Meaning	Code	Meaning	Code
*	0AAh	?	03Fh
=	0BDh	@	0C0h
#	0A3h	&	0A6h
[	0DBh	_	05Fh
]	0DDh	O	0CFh

(2) When the EIA code is used or when the ISO code is used with bit 4 (ISO) of parameter No. 6008 set to 1

Meaning	Code	Meaning	Code
*	Code set in parameter No. 6010	?	Code set in parameter No. 6015
=	Code set in parameter No. 6011	@	Code set in parameter No. 6016
#	Code set in parameter No. 6012	&	Code set in parameter No. 6017
[	Code set in parameter No. 6013	_	Code set in parameter No. 6018
]	Code set in parameter No. 6014		

For O, the same code as for O indicating a program number is used. Set a hole pattern for each of *, =, #, [,], ?, @, &, and _ in the ISO or EIA code in the corresponding parameters Nos. 6010 to 6018.
The code 00h cannot be used. The code indicating an alphabetic character can be used for the code indicating a character listed above, but the code can be no longer used to indicate the original character.

- Reserved words

The following reserved words are used in custom macros:

AND, OR, XOR, MOD, EQ, NE, GT, LT, GE, LE,
SIN, COS, TAN, ASIN, ACOS, ATAN, ATN, SQRT, SQR, ABS, BIN, BCD, ROUND, RND, FIX, FUP, LN, EXP, POW, ADP, IF, GOTO, WHILE, DO, END, BPRNT, DPRNT, POPEN, PCLOS, SETVN, AX, AXNUM, PRM

System variable (constant) names and registered common variable names are also used as reserved words.

16.14 EXTERNAL OUTPUT COMMANDS

In addition to the standard custom macro commands, the following macro commands are available. They are referred to as external output commands.

- BPRNT
- DPRNT

- POPEN
- PCLOS

These commands are provided to output variable values and characters through the input/output interface. In the external output commands, RS232-C, Memory card, USB memory, Data Server, and Embedded Ethernet can be specified for an external input/output device by the parameter No.0020.

Explanation

Specify these commands in the following order:

Open command: POPEN

Before specifying a sequence of data output commands, specify this command to establish a connection to an external input/output device.

Data output command: BPRNT or DPRNT

Specify necessary data output.

Close command: PCLOS

When all data output commands have completed, specify PCLOS to release a connection to an external input/output device.

- Open command POPEN

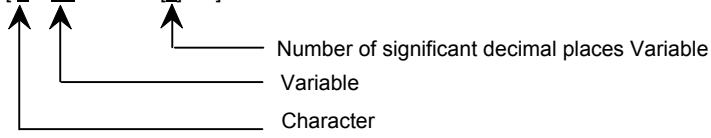
The POPEN command establishes a connection to an external input/output device. It must be specified before a sequence of data output commands.

NOTE
1 When RS232-C is used as an external input/output device and the specification number of the parameter No.102, 112 or 122 is set to 0, the CNC outputs a DC2 control code by POPEN command.
2 When Memory card, USB memory, Data Server, or Embedded Ethernet is used as an external input/output device, the file is opened by POPEN command. Please refer to the later explanation about the file name.

- Data output command BPRNT

The BPRNT command outputs characters and variable values in binary.

BPRNT [a #b [c] …]



(i) Specified characters are output.
Specifiable characters are as follows:
- Letters (A to Z)
- Numbers
- Special characters (*, /, +, -, ?, @, &, _)

NOTE
1 An asterisk (*) is output by a space code.
2 When specifying RS232-C as an external input/output device, specified characters are converted to the codes according to the setting data (ISO bit1 of parameter No.0000) that is output at that time. When using ?, @, &, and/or _, use the ISO code as the output code (setting data (ISO) = 1).

(ii) All variables are stored with a decimal point. Specify a variable followed by the number of significant decimal places enclosed in brackets. A variable value is treated as 2-word (32-bit) data, including the decimal digits. It is output as binary data starting from the highest byte.
(iii) When specified data has been output, an EOB code is output.

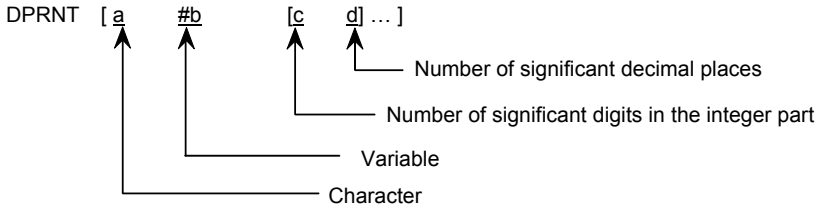
NOTE
When specifying RS232-C as an external input/output device, specified EOB code is converted to the codes according to the setting data (ISO bit1 of parameter No.0000) that is output at that time.

(iv) <Null> variables are regarded as 0.

Example

```
BPRNT [ C** X#100 [3] Y#101 [3] M#10 [0] ]
      Variable value
        #100=0.40956
        #101=-1638.4
        #10=12.34
      are output as follows:
C3 A0 A0 D8 00 00 01 9A 59 FF E7 00 00 4D 00 00 00 0C 0A
↓        ↓           ↓           ↓           ↓        ↓
C  sp sp  X0000019A    YFFE70000   M0000000C   LF
     ( * * )  (410)       (-1638400)    (12)      ( ; )
```

- Data output command DPRNT



The DPRNT command outputs characters and each digit in the value of a variable.
(i) For an explanation of the DPRNT command, see Items (i), (iii), and (iv) for the BPRNT command.
(ii) When outputting a variable, specify # followed by the variable number, then specify the number of digits in the integer part and the number of decimal places enclosed in brackets.
For the value of a variable, as many codes as the specified number of digits are output according to the settings one by one, starting with the highest digit. The decimal point is also output using the set code.
Each variable must be a numeric value consisting of up to nine digits. When high-order digits are zeros, these zeros are not output if bit 1 (PRT) of parameter No. 6001 is 1. If parameter PRT is 0, a space code is output each time a zero is encountered.
When the number of decimal places is not zero, digits in the decimal part are always output. If the number of decimal places is zero, no decimal point is output.
When bit 1 (PRT) of parameter No. 6001 is 0, a space code is output to indicate a positive number instead of +; if parameter PRT is 1, no code is output.

Example

DPRNT [X#2 [53] Y#5 [53] T#30 [20]]

Variable value

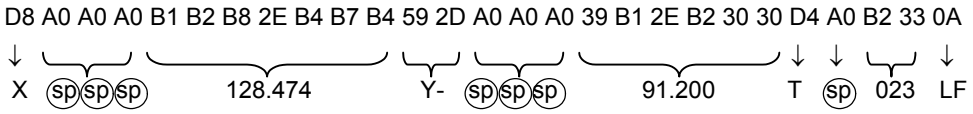
#2=128.47398

#5=-91.2

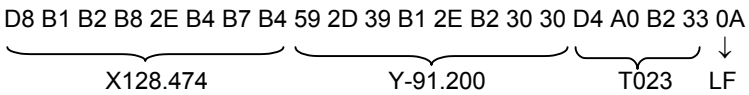
#30=123.456

are output as follows:

(1) Bit 1 (PRT) of parameter No. 6001 = 0



(2) Bit 1 (PRT) of parameter No. 6001 = 1



- Close command PCLOS

The PCLOS command releases a connection to an external input/output device. Specify this command when all data output commands have terminated.

NOTE

1 When RS232-C is used as an external input/output device and the specification number of the parameter No.102, 112 or 122 is set to 0, the CNC outputs a DC4 control code by PCLOS command.

2 When Memory card, USB memory, Data Server, or Embedded Ethernet is used as an external input/output device, the file is closed by PCLOS command.

File name

When Memory card, USB memory, Data Server, or Embedded Ethernet is used as an external input/output device, the created file name is from “PRNT0000.DAT” to “PRNT9999.DAT”. The numerical part of the file name is sequentially counted from 0000 to 9999. When the CNC is turned off, the number is counted from 0 again. But the number can be memorized and continuously counted by the bit 7 (SFN) of parameter No.6019.

Moreover, the created file name can be fixed as “MCR_PRNT.TXT” by the bit 3 (OFN) of parameter No.6019.

NOTE

1 If the file which has the same file name already exists when the file is created into an external input/output device, the following operation is done by an external input/output device.

In case of memory card : Alarm SR1973 is issued.
In case of USB memory : The file is overwritten.
In case of Data Server (Storage mode) : Alarm PS2032 is issued.
In case of Data Server (FTP mode) : The file is overwritten. (NOTE2)
In case of Embedded Ethernet : The file is overwritten. (NOTE2)

If a file with the same name already exists, the external output command cannot use bit 1 (COW) of parameter No. 11308, for whether to overwrite the file or cancel overwriting, cannot be used.

NOTE
2 This operation depends on the specification of FTP server software on the Host computer. Generally, the file is overwritten. Please refer to the specification of the FTP server software for details.

- Notice

NOTE
1 When an external input/output device is RS232-C, it is necessary to set the settings (baud-rate etc.) for RS232-C interface. Moreover, set 0 or 4 to the specification number of the input/output device of the parameters No, 102, 112 and 122.
2 It is not necessary to always specify the open command (POPEN), data output command (BPRNT, DPRNT), and close command (PCLOS) together. Once an open command is specified at the beginning of a program, it does not need to be specified again except after a close command was specified.
3 Be sure to specify open commands and close commands in pairs. Specify the close command at the end of the program. However, do not specify a close command if no open command has been specified.
4 When a reset operation is performed while commands are being output by a data output command, output is stopped and subsequent data is erased. Therefore, when a reset operation is performed by a code such as M30 at the end of a program that performs data output, specify a close command at the end of the program so that processing such as M30 is not performed until all data is output.
5 When an external input/output device is Data Server (FTP mode) or Embedded Ethernet, the FTP is used. Therefore, it is necessary to execute the FTP server on the personal computer.
Moreover, please note that there is a possibility that the FTP communication is aborted by the security function of the personal computer side when a time that the data doesn't flow is long.

16.15 RESTRICTIONS

- Sequence number search

A custom macro program cannot be searched for a sequence number.

- Single block

Even while a macro program is being executed, blocks can be stopped in the single block mode.
A block containing a macro call command (G66, G66.1, Ggg, Mmm, or G67) does not stop even when the single block mode is on.
Whether blocks containing arithmetic and logic operation commands and control commands are stopped depends on the settings of bits 5 (SBM) and 7 (SBV) of parameter No. 6000 as shown in the following table.

<table>
<tr><th colspan="2" rowspan="2"></th><th colspan="2">Bit 5 (SBM) of parameter No. 6000</th></tr>
<tr><th>0</th><th>1</th></tr>
<tr><td rowspan="2">Bit 7 (SBV) of parameter No. 6000</td><td>0</td><td>Not stopped when the single block mode is on.</td><td rowspan="2">Can be stopped in the single block mode. (Variable #3003 cannot be used to disable single block stop. Single block stop is always enabled.)</td></tr>
<tr><td>1</td><td>Can be stopped in the single bock mode. (Variable #3003 can be used to enable or disable single block stop.)</td></tr>
</table>

Note that when a single block stop occurs at a macro statement in cutter compensation mode, the statement is assumed to be a block that does not involve movement, and proper compensation cannot be performed in some cases. (Strictly speaking, the block is regarded as specifying a movement with a travel distance 0.)

- Optional block skip

A / appearing in the middle of an <expression> (enclosed in brackets [ ] on the right-hand side of an arithmetic expression) is regarded as a division operator; it is not regarded as the specifier for an optional block skip code.

- Operation in EDIT mode

By setting bit 0 (NE8) of parameter No.3202 and bit 4 (NE9) of parameter No.3202 to 1, deletion and editing are disabled for custom macro programs and subprograms with program numbers 8000 to 8999 and 9000 to 9999. This prevents registered custom macro programs and subprograms from being destroyed by accident. When the entire memory is cleared, the contents of memory such as custom macro programs are deleted.

- Reset

With a reset operation, local variables and common variables #100 to #199 are cleared to null values. They can be prevented from clearing by setting bit 6 (CCV) of parameter No.6001. System variables #100 to #199 are not cleared.
A reset operation clears any called states of custom macro programs and subprograms, and any DO states, and returns control to the main program.

- Display of the PROGRAM RESTART

As with M98, the M and T codes used for subprogram calls are not displayed.

- Feed hold

When a feed hold is enabled during execution of a macro statement, the machine stops after execution of the macro statement. The machine also stops when a reset or alarm occurs.

- DNC operation

The control commands (such as GOTO and WHILE-DO) cannot be executed during DNC operation.
However, this restriction is removed when a program registered in program memory is called during DNC operation.

- Constant values that can be used in <expression>

+0.00000000001 to +999999999999
-999999999999 to -0.00000000001
The number of significant digits is 12 (decimal).
If this range is exceeded, an alarm PS0004, “INVALID BREAK POINT OF WORDS” occurs.

- Comment section

Usually, the position in which comment section can be inserted in macro statements is as follows.

```
(ABC) #100 = 1;          Head of the block
#100 = 1(ABC);           End of the block
N01 (ABC) #100 = 1;      Just behind of the sequence number
```

If the parameter NCM(No.6020#3) is set to 1, the position in which comment section can be inserted in macro statements as follows, too.

```
#100(ABC) = 1;                    Just behind of variable number
#100 = #101 +1.(ABC) *#102;       Just behind of numeric values
#100 = [#_UIL[1]](ABC) *100.;     Just behind of the name of variable
```

In word editing, it might become difficult to do the program edit since comment section is inserted. Such a case uses character editing.
Comment section cannot be inserted in the character string such as variable number, numeric values, name of variable, functions and control commands.
Example that cannot be inserted

```
#(ABC)100 = 1;
#100 = [#_UIL(ABC)[1]] *100.;
IF[ #100 EQ 1 ] GO(ABC)TO99;
```

16.16 INTERRUPTION TYPE CUSTOM MACRO

When a program is being executed, another program can be called by inputting an interrupt signal (UINT) from the machine.
This function is referred to as an interruption type custom macro function. Program an interrupt command in the following format:

Format

M96Pxxxxxxxx ;	Enables custom macro interrupt
M97 ;	Disables custom macro interrupt

Explanation

Use of the interruption type custom macro function allows the user to call a program during execution of an arbitrary block of another program. This allows programs to be operated to match situations which vary from time to time.

(1) When a tool abnormality is detected, processing to handle the abnormality is started by an external signal.
(2) A sequence of machining operations is interrupted by another machining operation without the cancellation of the current operation.
(3) At regular intervals, information on current machining is read.

:

Listed above are examples like adaptive control applications of the interruption type custom macro function.

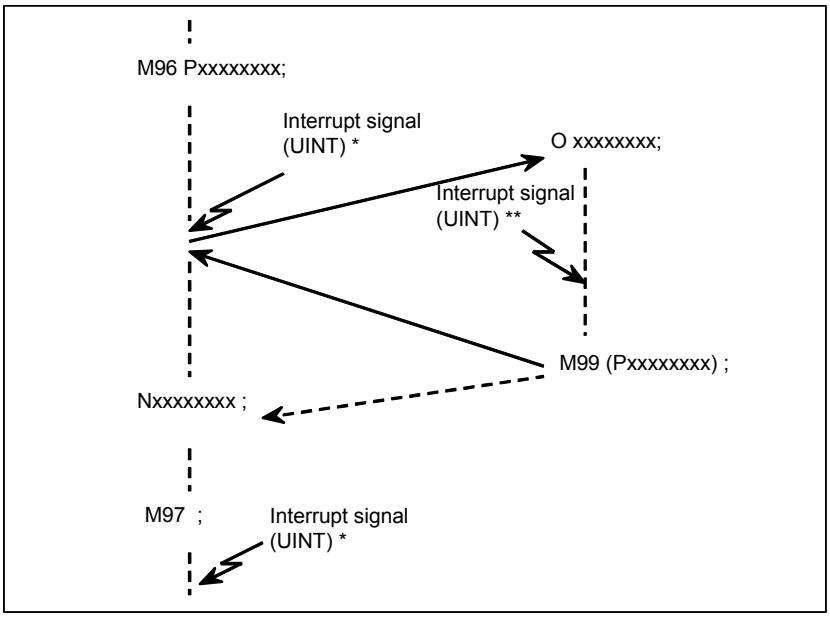


Fig 16.16 (a) Interruption type custom macro function

When M96Pxxxx is specified in a program, subsequent program operation can be interrupted by an interrupt signal (UINT) input to execute the program specified by Pxxxx. When the interrupt signal (UINT, marked with an asterisk (*) in Fig 16.16 (a)) is input during execution of the interrupt program or after M97, it is ignored.

16.16.1 Specification Method

Explanation

- Interrupt conditions

A custom macro interrupt is available only during program execution. It is enabled under the following conditions

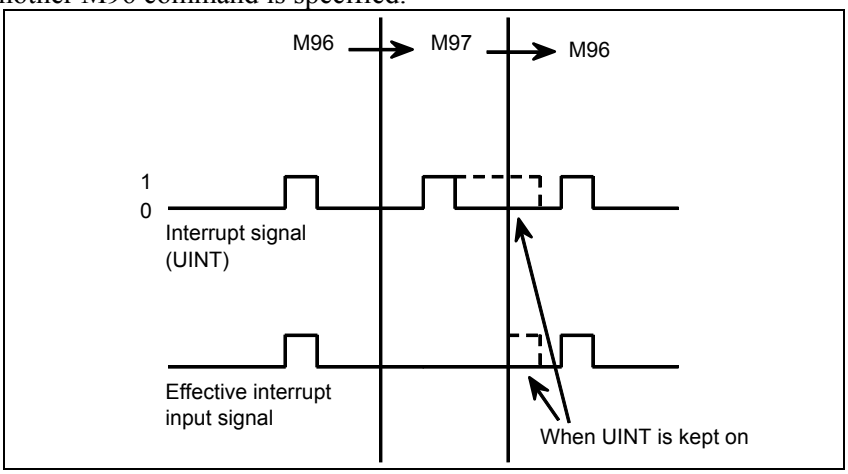
- When memory operation, DNC operation, or MDI operation is selected
- When STL (start lamp) is on
- When a custom macro interrupt is not currently being processed

A macro interrupt cannot be performed during manual operation.

- Specification

Generally, the custom macro interrupt function is used by specifying M96 to enable the interrupt signal (UINT) and M97 to disable the signal.

Once M96 is specified, a custom macro interrupt can be initiated by the input of the interrupt signal (UINT) until M97 is specified or the CNC is reset. After M97 is specified or the CNC is reset, no custom macro interrupts are initiated even when the interrupt signal (UINT) is input. The interrupt signal (UINT) is ignored until another M96 command is specified.



The interrupt signal (UINT) becomes valid after M96 is specified. Even when the signal is input in M97 mode, it is ignored. When the signal input in M97 mode is kept on until M96 is specified, a custom macro interrupt is initiated as soon as M96 is specified (only when the status-triggered scheme is employed); when the edge-triggered scheme is employed, the custom macro interrupt is not initiated even when M96 is specified.

NOTE
For the status-triggered and edge-triggered schemes, see the Subsection, “Details of Functions”.

16.16.2 Details of Functions

Explanation

- Subprogram-type interrupt and macro-type interrupt

There are two types of custom macro interrupts: Subprogram-type interrupts and macro-type interrupts. The interrupt type used is selected by bit 5 (MSB) of parameter No.6003.

(a) Subprogram-type interrupt: When bit 5 (MSB) of parameter No.6003 is set to 1
An interrupt program is called as a subprogram.
This means that the levels of local variables remain unchanged before and after the interrupt.
This interrupt is not included in the nesting level of subprogram calls.

(b) Macro-type interrupt: When bit 5 (MSB) of parameter No.6003 is set to 0
An interrupt program is called as a custom macro.
This means that the levels of local variables change before and after the interrupt.
The interrupt is not included in the nesting level of custom macro calls.
When a subprogram call or a custom macro call is performed within the interrupt program, this call is included in the nesting level of subprogram calls or custom macro calls.
Arguments cannot be passed from the current program even when the custom macro interrupt is a macro-type interrupt. The local variables immediately after interruption are all cleared to null.

- M codes for custom macro interrupt control

In general, custom macro interrupts are controlled by M96 and M97. However, these M codes, may already being used for other purposes (such as an M function or macro M code call) by some machine tool builders.
For this reason, bit 4 (MPR) of parameter No.6003 is provided to set M codes for custom macro interrupt control.
When specifying this parameter to use the custom macro interrupt control M codes set by parameters, set parameters Nos.6033 and 6034 as follows:
Set the M code to enable custom macro interrupts in parameters Nos.6033, and set the M code to disable custom macro interrupts in parameter 6034.
When specifying that parameter-set M codes are not used, M96 and M97 are used as the custom macro control M codes regardless of the settings of parameters Nos.6033 and 6034.
The M codes used for custom macro interrupt control are processed internally (they are not output to external units). However, in terms of program compatibility, it is undesirable to use M codes other than M96 and M97 to control custom macro interrupts.

- Custom macro interrupts and NC statements

When performing a custom macro interrupt, the user may want to interrupt the NC statement being executed, or the user may not want to perform the interrupt until the execution of the current block is completed. Bit 2 (MIN) of parameter No.6003)is used to select whether to perform interrupts even in the middle of a block or to wait until the end of the block. The type of interrupt performed even in the middle of a block is called type I and the type of interrupt performed at the end of the block is called type II.

⚠ CAUTION

For interrupt type I, operation after control is returned differs depending on whether the interrupt program contains an NC statement.
When the program number block contains EOB (;), it is assumed to contain an NC statement.

(Program containing an NC statement)

```
O0013;
#101=#5041;
#102=#5042;
#103=#5043;
M99;
```

(Program containing no NC statement)

```
O0013#101=#5041;
#102=#5042;
#103=#5043;
M99;
```

Type I (when an interrupt is performed even in the middle of a block)

(i) When the interrupt signal (UINT) is input, any movement or dwell being performed is stopped immediately and the interrupt program is executed.

(ii) If there are NC statements in the interrupt program, the command in the interrupted block is lost and the NC statement in the interrupt program is executed. When control is returned to the interrupted program, the program is restarted from the next block after the interrupted block.

(iii) If there are no NC statements in the interrupt program, control is returned to the interrupted program by M99, then the program is restarted from the command in the interrupted block.

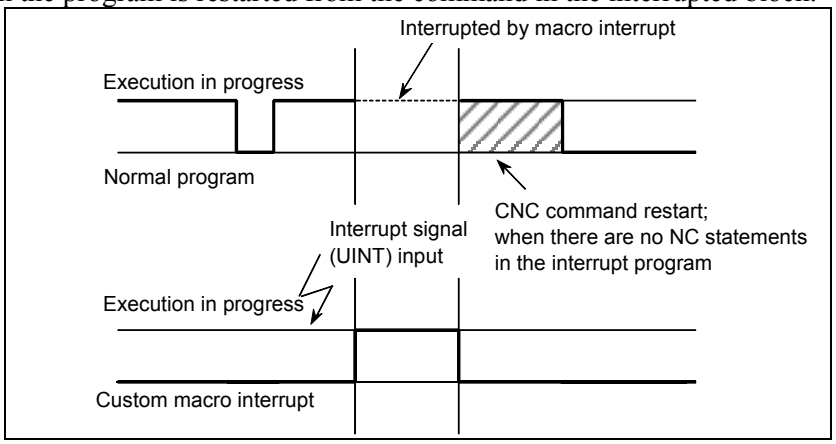


Fig. 16.16 (b) Custom macro interrupt and NC command (type I)

Type II (when an interrupt is performed at the end of the block)

(i) If the block being executed is not a block that consists of several cycle operations such as a drilling canned cycle and automatic reference position return (G28), an interrupt is performed as follows:
When an interrupt signal (UINT) is input, macro statements in the interrupt program are executed immediately unless an NC statement is encountered in the interrupt program. NC statements are not executed until the current block is completed.

(ii) If the block being executed consists of several cycle operations, an interrupt is performed as follows:
When the last movement in the cycle operations is started, macro statements in the interrupt program are executed unless an NC statement is encountered. NC statements are executed after all cycle operations are completed.

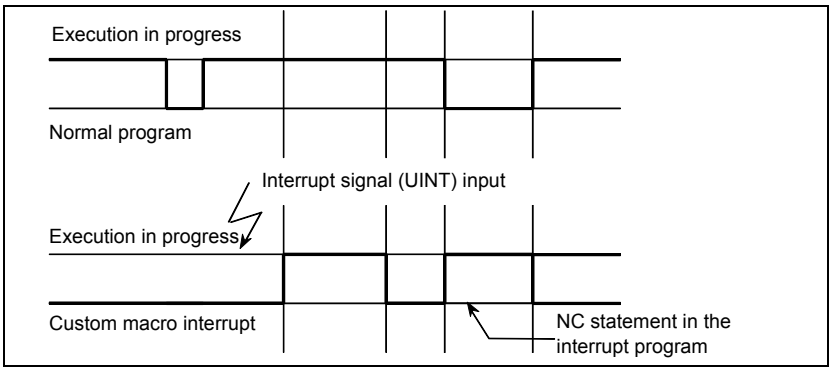


Fig. 16.16 (c) Custom macro interrupt and NC command (type II)

M

NOTE

During execution of a program for cycle operations, interrupt type II is performed regardless of whether bit 2 (MIN) of parameter No. 6003 is set to 0 or 1. Cycle operations are available for the following functions:

<1> Automatic reference position return
<2> Tool radius · tool nose radius compensation (generating multiple blocks using the specified block such as when the tool moves around the outside of an acute angle)
<3> Canned cycle
<4> Automatic tool length measurement
<5> Optional chamfering/corner R
<6> Exponential interpolation
<7> Normal direction control
<8> Cutting point interpolation for cylindrical interpolation

T

NOTE

During execution of a program for cycle operations, interrupt type II is performed regardless of whether bit 2 (MIN) of parameter No. 6003 is set to 0 or 1. Cycle operations are available for the following functions:

<1> Automatic reference position return
<2> Tool radius · tool nose radius compensation (generating multiple blocks using the specified block such as when the tool moves around the outside of an acute angle)
<3> Canned cycle (No interruption type custom macro can be used during execution of a multiple repetitive canned turning cycle, however.)
<4> Automatic tool length measurement
<5> Chamfering/corner R
<6> Exponential interpolation
<7> Cutting point interpolation for cylindrical interpolation

- Conditions for enabling and disabling the custom macro interrupt signal

The interrupt signal becomes valid after execution starts of a block that contains M96 for enabling custom macro interrupts. The signal becomes invalid when execution starts of a block that contains M97.

While an interrupt program is being executed, the interrupt signal becomes invalid. The signal become valid when the execution of the block that immediately follows the interrupted block in the main program is started after control returns from the interrupt program. In type I, if the interrupt program consists of only macro statements, the interrupt signal becomes valid when execution of the interrupted block is started after control returns from the interrupt program.

- Custom macro interrupt signal (UINT)

There are two schemes for custom macro interrupt signal (UINT) input: The status-triggered scheme and edge-triggered scheme. When the status-triggered scheme is used, the signal is valid when it is on. When the edge triggered scheme is used, the signal becomes valid on the rising edge when it switches from off to on status.
One of the two schemes is selected with bit 3 (TSE) of parameter No.6003.

When the status-triggered scheme is selected by this parameter, a custom macro interrupt is generated if the interrupt signal (UINT) is on at the time the signal becomes valid. By keeping the interrupt signal (UINT) on, the interrupt program can be executed repeatedly.

When the edge-triggered scheme is selected, the interrupt signal (UINT) becomes valid only on its rising edge. Therefore, the interrupt program is executed only momentarily (in cases when the program consists of only macro statements). When the status-triggered scheme is inappropriate, or when a custom macro interrupt is to be performed just once for the entire program (in this case, the interrupt signal may be kept on), the edge-triggered scheme is useful.

Except for the specific applications mentioned above, use of either scheme results in the same effects. The time from signal input until a custom macro interrupt is executed does not vary between the two schemes.

In the example shown in Fig. 16.16 (d), an interrupt is executed four times when the status-triggered scheme is used; when the edge-triggered scheme is used, the interrupt is executed just once.

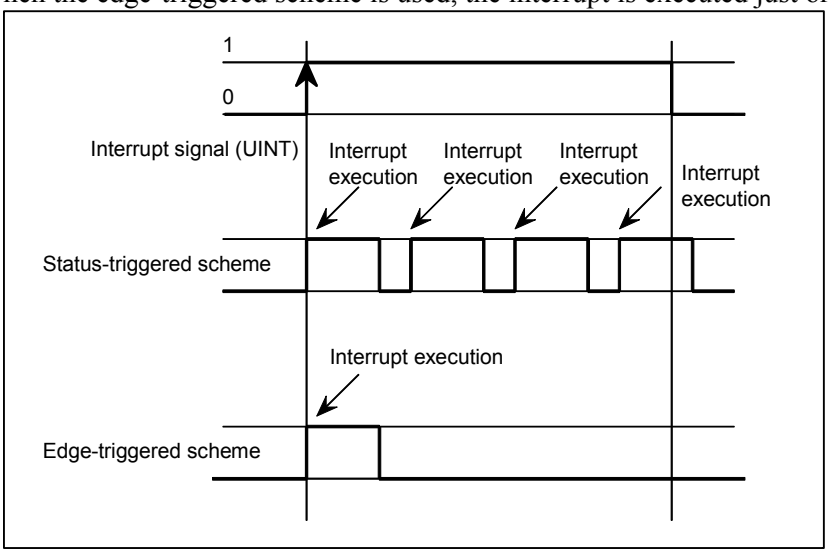


Fig. 16.16 (d) Custom macro interrupt signal

- Return from a custom macro interrupt

To return control from a custom macro interrupt to the interrupted program, specify M99. A sequence number in the interrupted program can also be specified using address P. If this is specified, the program is searched from the beginning for the specified sequence number. Control is returned to the first sequence number found.

NOTE
When an M99 block consists only of address O, N, P, L, or M, this block is regarded as belonging to the previous block in the program. Therefore, a single-block stop does not occur for this block. In terms of programming, the following <1> and <2> are basically the same. (The difference is whether Gxx is executed before M99 is recognized.)
<1> Gxx Xxxx ;
M99 ;
<2> Gxx Xxxx M99 ;

- Custom macro interrupt and modal information

A custom macro interrupt is different from a normal program call. It is initiated by an interrupt signal (UINT) during program execution. In general, any modifications of modal information made by the interrupt program should not affect the interrupted program.
For this reason, even when modal information is modified by the interrupt program, the modal information before the interrupt is restored when control is returned to the interrupted program by M99.

When control is returned from the interrupt program to the interrupted program by M99 Pxxxxxxxx, however, modal information can again be controlled by the program. In this case, the new continuous information modified by the interrupt program is passed to the interrupted program.

In this case, take the following action as required:
<1> The interrupt program provides modal information to be used after control is returned to the interrupted program.
<2> After control is returned to the interrupted program, modal information is specified again as necessary.

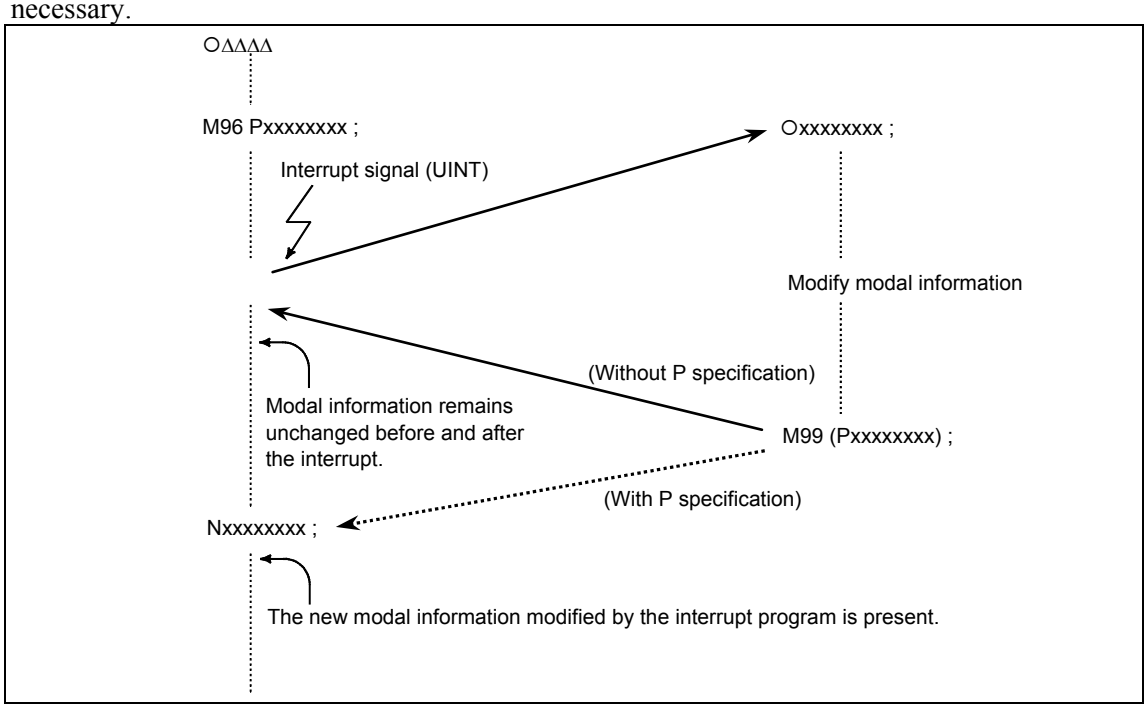


Fig. 16.16 (e) Custom macro interrupt and modal information

Modal information when control is returned by M99

The modal information present before the interrupt becomes valid. The new modal information modified by the interrupt program is made invalid.

Modal information when control is returned by M99 Pxxxxxxxx

The new modal information modified by the interrupt program remains valid even after control is returned.

Modal information which was valid in the interrupted block

The old modal information which was valid in the interrupted block can be read using custom macro system variables #4401 to #4530.

M

System variable	Modal information which was valid when a custom macro interrupt was generated
#4401	G code (group 01)
:	:
#4421	G code (group 21)
#4502	B code
#4507	D code
#4508	E code
#4509	F code
#4511	H code
#4513	M code
#4514	Sequence number
#4515	Program number
#4519	S code
#4520	T code
#4530	Additional workpiece coordinate system number

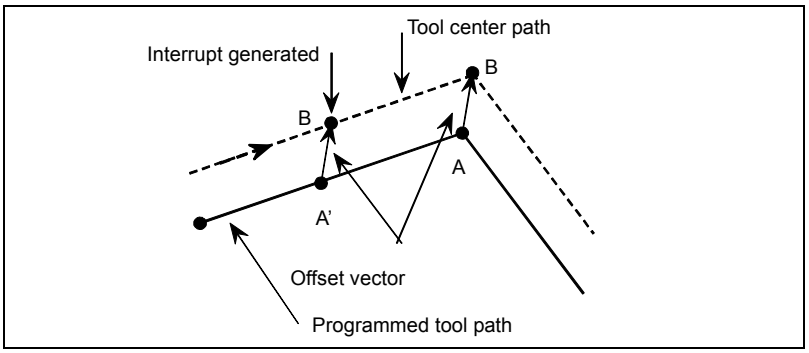
T

System variable	Modal information which was valid when a custom macro interrupt was generated
#4401	G code (group 01)
:	:
#4421	G code (group 21)
#4508	E code
#4509	F code
#4513	M code
#4514	Sequence number
#4515	Program number
#4519	S code
#4520	T code
#4530	Additional workpiece coordinate system number

- System variables (position information values) for the interrupt program

Position information can be read as follows.

Macro variable	Condition	Position information value
#5001 or above	Until the first NC statement appears	Coordinates of point A
	After an NC statement with no move command appears	Coordinates of point A'
	After an NC statement with a move command appears	Coordinates of the end point of the move command
#5021 or above		Machine coordinates of point B'
#5041 or above		Workpiece coordinates of point B'



- Custom macro interrupt and custom macro modal call

When the interrupt signal (UINT) is input and an interrupt program is called, the custom macro modal call is canceled (G67). However, when G66 is specified in the interrupt program, the custom macro modal call becomes valid. When control is returned from the interrupt program by M99, the modal call is restored to the state it was in before the interrupt was generated. When control is returned by M99 Pxxxxxxxx;, the modal call in the interrupt program remains valid.

- Custom macro interrupt and program restart

In program restart, when the interrupt signal (UINT) is input during dry run recovery after a search, the interrupt program is called after restart of all axes is completed.
That is, interrupt type II is assumed regardless of the parameter setting.

M

NOTE
1 Alarm PS1101, “ILLEGAL CNC STATEMENT IRT.” occurs in the following cases:
<1>An interrupt is generated in the programmable mirror image (G51.1) mode and another G51.1 is specified in the interrupt program.
<2>An interrupt is generated in the coordinate system rotation (G68) mode and another G68 is specified in the interrupt program.
<3>An interrupt is generated in the scaling (G51) mode and another G51 is specified in the interrupt program.
2 In program restart, do not input the interrupt signal (UINT) during dry run recovery after a search.

T

NOTE
1 Alarm PS1101, “ILLEGAL CNC STATEMENT IRT.” occurs in the following cases:
<1>An interrupt is generated in the programmable mirror image (G51.1) mode and another G51.1 is specified in the interrupt program.
<2>An interrupt is generated in the coordinate system rotation (G68.1) mode and another G68.1 is specified in the interrupt program.
<3>An interrupt is generated in the scaling (G51) mode and another G51 is specified in the interrupt program.
2 No interruption type custom macro can be used during execution of a multiple repetitive canned turning cycle.
3 In program restart, do not input the interrupt signal (UINT) during dry run recovery after a search.

17 REAL-TIME CUSTOM MACRO

Chapter 17, "REAL-TIME CUSTOM MACRO", consists of the following sections:

17.1 TYPES OF REAL TIME MACRO COMMANDS 503
17.2 VARIABLES 508
17.3 ARITHMETIC AND LOGICAL OPERATION 514
17.4 CONTROL ON REAL TIME MACRO COMMANDS 515
17.5 MACRO CALL 521
17.6 OTHERS 522
17.7 AXIS CONTROL COMMAND 523
17.8 NOTES 533
17.9 LIMITATION 535

Overview

Used with an NC program, the real time custom macro function controls peripheral axes and signals.

If a macro statement is used together with an NC statement, a program using the conventional custom macro function executes the macro statement immediately when the macro statement is read. So, the macro statement cannot be executed independently of the NC statement.
On the other hand, the real time custom macro function enables the following control operations when a real time macro command (RTM command) is coded in an NC program.

- A real time macro command starts operation in synchronism with the NC statement and is executed independently. During NC program execution, a real time macro command can be executed at the same time.
- PMC interface signals can be read and written (with a restriction). In an NC program, a motion using a signal as a trigger can be coded.
- Variables dedicated to a real time macro command can be read and written.
- A real time macro command can exercise axis control. (The option is required.)
- Multiple real time macro commands can be executed at the same time. Multiple real time custom macro statements can be coded in an NC program and can be controlled independently of each other.

By using a real time custom macro, control on signals and peripheral axes can be programmed.
The example below turns on and off a signal and performs operation on a peripheral axis when machining points are passed.

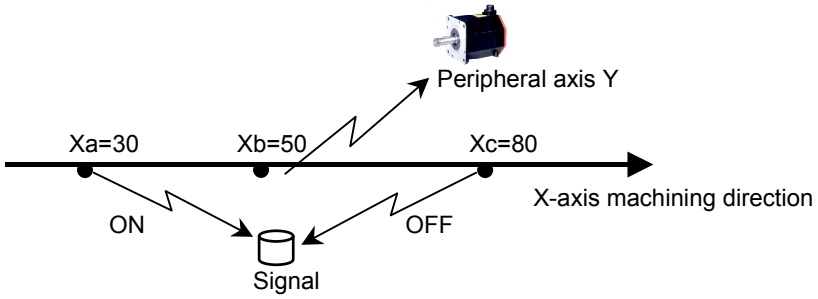


Fig. 17 (a) Signal and operation on a peripheral axis during machining (example)

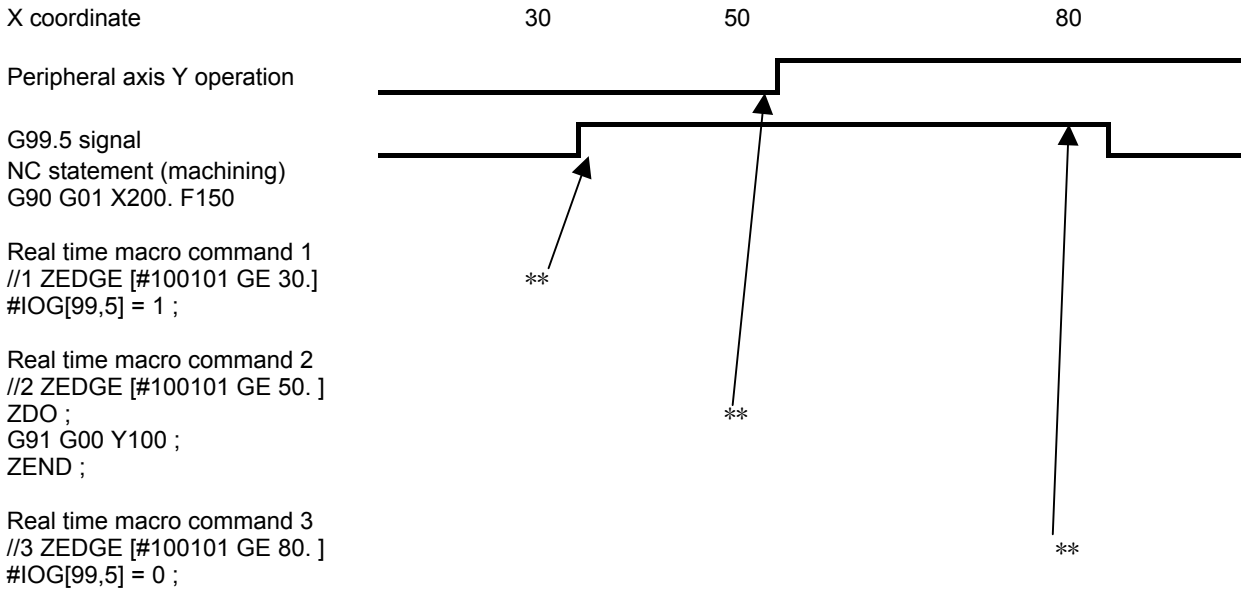
During movement for machining along the X-axis in the Fig. 17 (a):
<1> When point Xa is passed, the G99.5 signal is set to 1.
→ Real time macro command 1
<2> When point Xb is passed, positioning is started on peripheral axis Y. → Real time macro command 2
<3> When point Xc is passed, the G99.5 signal is set to 0.

→ Real time macro command 3

The operation above is programmed using real time macro commands.

```
Program
O0001 ;
G92 X0 ;
//1 ZEDGE [#100101 GE 30. ] #IOG[99,5] = 1 ;
//2 ZEDGE [#100101 GE 50.] ZDO ;
G91 G00 Y100 ;
ZEND ;
//3 ZEDGE [#100101GE 80. ] #IOG[99,5] = 0 ;
G90 G01 X200. F150 ;
M30 ;
```

The timing is as follows. (‘**’ represents the satisfaction of the condition.)



Explanation

To use the real time custom macro function, a real time macro command (RTM command) is coded in an NC program.

- Real time macro command (RTM command)

The real time macro command (RTM command) is a macro command that starts execution in phase with an NC statement in the program. After the execution of a real time macro command starts, the RTM command operates independently of the NC statement.
An RTM command is a statement dedicated to the real time custom macro function.
An RTM command consists of a set of one or more real time macro statements (RTM statements).

- Real time macro statement (RTM statement)

The real time macro statement (RTM statement) is a statement included in an RTM command.
One or more RTM statements make up an RTM command.
An RTM statement consists of a macro command and axis control command dedicated to the real time custom macro function.

The axis control command of an RTM statement is an RTM statement including an address. This command is used to exercise axis control.

Example
```
// ZDO ;
G91 G00 X100 ;
ZEND ;
```
(ZDO and ZEND are reserved words required for the axis control command of an RTM statement, and are detailed later.)

The macro command of an RTM statement is a macro statement used with an RTM statement for arithmetic/logical operation and signal control. (In the example below, #RV[1] and #RV[2] are variables dedicated to the real time macro command.)
Example
```
// #IOG[124, 5] = 1 ;
// #RV[1] = #RV[2] * 10 ;
```

Format

The real time macro command format is shown below.
The RTM command is a command with two slashes (//) prefixed at the start of a block.

//n <real-time-macro-statement> **or** **//n ZDO ;** **<real-time-macro-statement>** **:** **ZEND ;**
n: Modal ID (1 to 10) (Omissible) When a proper number is coded in n, a modal real time macro command is specified. When n is omitted, a one-shot real time macro command is specified.

ZDO to ZEND are detailed later.

17.1 TYPES OF REAL TIME MACRO COMMANDS

17.1.1 Modal Real Time Macro Command / One-shot Real Time Macro Command

Explanation

A command with '//' followed by an RTM statement is referred to as a one-shot real time macro command (one-shot RTM command).
Example:
```
// #RV[1] = 30 ;
```

On the other hand, a command with '//' followed by number n (1 to 10) then an RTM statement is referred to as a modal real time macro command (modal RTM command).
Example:
```
//3 #RV[1] = 30 ;
```

A one-shot RTM command starts when the execution of the first following NC command starts. A started one-shot RTM command is valid until the end of the NC command.
A modal RTM command starts when the execution of the first following NC command starts, as in the case of a one-shot RTM command.
Unlike a one-shot RTM command, however, a started modal RTM command is valid until the automatic operation is completed.

- Start of a real time macro command

An RTM command starts when the execution of the first following NC command starts.

Example:

When NC command (1) starts execution in the program below, macro commands (2) and (4) are executed in succession without waiting for the end of (1).
On the other hand, RTM command (3)starts execution when NC command (5) starts execution after the end of NC command (1).

```
O0001 ;
G90 G00 X30. ;          (1) NC command
#100=0 ;                (2) Macro command
// #RV[0]=1 ;           (3) RTM command
#102=2;                 (4) Macro command
G90 G00 X100. ;         (5) NC command
M30 ;
```

- End of a real time macro command

When one of the following conditions is satisfied, the RTM command is terminated.

Termination conditions common to one-shot RTM and modal RTM commands

- When RTM command processing is completed
- When a reset occurs

Termination condition specific to a one-shot RTM command

- When the execution of the NC command that started at the same time has ended

However, if the RTM statement being executed is an axis control command, the command is terminated when the execution of the block is completed.
If the execution of the Y10. block is completed before the X100. block in the command below, and the execution of the Y20. block starts, for example, the X100. command of the RTM statement is executed until the end.

```
// ZDO ;
X100. ;
ZEND ;
Y10. ;
Y20. ;
```

NOTE

1 No one-shot RTM command can be specified using any of the commands indicated below as a trigger. When using any of these commands as a trigger, use a modal RTM command.
- Command related to reference position return
- Command related to AI contour control
- Command related to canned cycles (rigid tapping, drilling cycle, etc.)
- Command related to cutter compensation
- Command related to tool length compensation
- Command related to automatic tool length measurement
- Command related to coordinate system rotation
  Command related to scaling
- Command related to programmable mirror image

NOTE

2 If an RTM command is specified using, as a trigger, a block such as a block specifying NURBS interpolation or a multiple repetitive canned cycle that does not necessarily pass the start point or end point of the command, operation can start or end at a point other than the start point and end point. Do not use such a block as a trigger.

3 Do not restart a program that includes an RTM command.

4 When an NC statement used as a trigger for an RTM command represents an auxiliary function, execution continues even if the FIN signal is awaited.
If the following program is executed, for example, the count-up operation of #RV[0] continues until the FIN signal of M55 is returned:

```
O0001 ;
// ZWHILE [1] #RV[0] = #RV[0]+1 ;
M55 ;
G91 X200. ;
   :
```

5 If M02 follows an RTM command, execution continues until a reset occurs, even when the program itself is terminated.
If the following program is executed, for example, the count-up operation of #RV[0] continues until a reset occurs:

```
O0001 ;
// ZWHILE [1] #RV[0] = #RV[0]+1 ;
M02 ;
```

- Priority of commands

If a modal RTM command and one-shot RTM command are specified at the same time, the modal RTM command is executed first.
If multiple modal RTM commands are specified at the same time, the commands are executed in ascending order of ID values.
No priority is applicable to one-shot RTM commands.
The order of execution is as follows:

Modal RTM command with ID value being 1
Modal RTM command with ID value being 2
:
:
Modal RTM command with ID value being 10
One-shot RTM command
:
One-shot RTM command



Example 1)
Priority of modal RTM commands

```
O0001 ;
//1 #RV[0]=1 ;
//3 #RV[0]=3 ;
//2 #RV[0]=2 ;
M02 ;
```

When the program above is executed, the RTM commands are executed in the following order:
#RV[0]=1

```
#RV[0]=2
#RV[0]=3
```
So, the value of #RV[0] is 3.

Example 2)

Priority of modal RTM commands and a one-shot RTM command

```
O0001 ;
//3 #RV[0]=3 ;
//1 #RV[0]=1 ;
// #RV[0]=10 ;
//5 #RV[0]=5 ;
M02 ;
```

When the program above is executed, the RTM commands are executed in the following order:

```
//1 #RV[0]=1
//3 #RV[0]=3
//5 #RV[0]=5
// #RV[0]=10
```
So, the value of #RV[0] is 10.

Example 3)

Priority of one-shot RTM commands

When the following program is executed, the value of #RV[0] is undefined, that is, 1, 2, or 3.

```
O0001 ;
//#RV[0]=1 ;
//#RV[0]=2 ;
//#RV[0]=3 ;
G04P10 ;
M30 ;
```

Even if a priority is applicable, the order of execution or the order of termination can change when an RTM statement includes a control code, ZWHILE or ZEDGE, or axis control command.

Example 4)

The RTM command priority of #RV[0]=1 in a modal command with its ID value being 1 is higher than the priority of #RV[1]=1 in a modal command with its ID value being 2. However, #RV[0]=1 is executed after the end of the execution of the block specifying the axis control command G91 G00 X10., so that #RV[1]=1 is actually executed earlier than #RV[0]=1.

```
O0001 ;
//1 ZDO ;
G91 G00 X10. ;
#RV[0]=1 ;
ZEND ;
//2 #RV[1]=1 ;
G04 P10 ;
M30 ;
```

Example 5)

In the RTM command priority, ZEDGE in a modal command with its ID value being 1 is always a false control code (detailed later). The RTM command priority of #RV[0]=1 in a modal command with its ID value being 1 is higher than the priority of #RV[1]=1 and #RV[2]=1 in a modal command with their ID values being 2. However, #RV[0]=1 is executed after the condition of ZEDGE becomes true (that is, at the second time or later), so that #RV[1]=1 and #RV[2]=1 are executed earlier than #RV[0]=1.

```
O0001 ;
```

```
//1 ZEDGE [ #IOG[234.0] EQ 1 ] #RV[0]=1 ;
//2 ZDO ;
#RV[1]=1 ;
#RV[2]=1 ;
ZEND ;
G04 P10 ;
M30 ;
```

- Number of real time macro commands

A program can have multiple RTM commands coded.
Up to six one-shot RTM commands can be specified.
If one-shot RTM commands more than the maximum allowable number are specified, an alarm is issued.

Up to ten modal RTM commands can be specified. When specifying modal RTM commands, ensure that there is no duplicate ID.

If there is a duplicate ID, or an incorrect ID is specified, an alarm is issued.

In all paths, up to 16 RTM commands can be executed simultaneously.
When an axis control command is included, up to four commands can be executed simultaneously.

NOTE

1 In a block specifying an RTM statement, no NC command can be coded.
2 If the maximum specifiable number of commands or the maximum number of simultaneously executable commands is exceeded, a PS alarm is issued.
3 If an NC statement to be used to trigger an RTM command is specified in a block (e.g., small block) that ends in a very short time, an RTM statement programmed to start at a different timing may be executed simultaneously. If the following is specified, for example, #RV[0]=1 and #RV[1]=2 may be executed simultaneously:

```
// #RV[0]=1 ;
G91 G01 X0.002 ; F5000
// #RV[1]=2 ;
X0.001 ;
```

4 When a function for reading multiple blocks in advance is used, the same ID must not be coded in the scope of blocks read in advance. In the following program, for example, while //1 #RV[0]=#100101 is being executed during AI contour control, //1 #RV[1]=#100101 and //1 #RV[2]=#100101 are also read in advance. So, an alarm may be issued.

```
//1 #RV[0]=#100101 ;
X#100 ;
//1 #RV[1]=#100101 ;
X#100 ;
//1 #RV[2]=#100101 ;
X#100 ;
```

NOTE

5 When a function for reading multiple blocks in advance is used, up to three blocks among the blocks read in advance can trigger an RTM command. For example, if the blocks up to the block of (2) are read in advance during execution of (1) in the program below, up to three blocks can trigger an RTM command. In the program below, the number of NC blocks that trigger an RTM command exceeds 3, so that the RTM command of (a) must not be coded.

```
X30. Y50. ; ..... (1)
// Z-30. ;
// #RV[0]=#RV[0]+1 ;
X3. Y16. ;              ← NC block 1 triggering an RTM command
X-23. Y4. ;
// #RV[1]=#RV[1]+1 ;
//2 Z30. ;
X-2. Y9. ;              ← NC block 2 triggering an RTM command
X17. Y5. ;
// #RV[2]=#RV[2]+1 ;
X-2. Y9. ;              ← NC block 3 triggering an RTM command
// #RV[3]=#RV[3]+1 ; ... (a)
X-12. Y-3. ;
X-100. Y200. ; ..... (2)
```

- Reserved words

The following reserved words are used with real time custom macros:

- Reserved words dedicated to real time custom macros
  ZDO, ZEND, ZONCE, ZWHILE, ZEDGE
- Reserved words shared with custom macros
  AND, OR, XOR, MOD, EQ, NE, GT, LT, GE, LE, SIN, COS, TAN, ASIN, ACOS, ATAN, ATN, SQRT, SQR, ABS, BIN, BCD, ROUND, RND, FIX, FUP, LN, EXP, POW

Be sure to fully spell out the reserved words for real time custom macros. For example, ‘ZONCE’ must not be coded as ‘ZON’ or ‘ZONC’.

17.2 VARIABLES

Overview

With real time custom macros, the following variables can be handled:

- System variables dedicated to real time custom macros
- Variables (RTM variables) dedicated to real time custom macros
- System variables for some custom macros

The RTM variables mean the variables dedicated to real time custom macros.

Table 17.2 (a) List of usable variables

		Real time custom macro	Custom macro
Real time custom macro variables	System variables	Usable	Unusable
	RTM variables	Usable	Unusable
Custom macro variables	System variables	Partially usable	Usable
	Common variables	Unusable	Usable
	Local variables	Unusable	Usable

The variables (system variables and RTM variables) dedicated to real time custom macros are the variables specific to the real time custom macro function. Those variables cannot be used with the custom macro function.

NOTE
Real time custom macros do not support <null>. Do not use it.

17.2.1 Variables Dedicated to Real Time Custom Macros

These variables are dedicated to real time custom macros. The variables are classified as system variables and RTM variables.

17.2.1.1 System variables

System variables dedicated to real time custom macros

Format

#IOp [m, n] Bit-by-bit read/write
#IOpB [m] Byte-by-byte read/write
#IOpW [m] Word-by-word read/write
#IOpD [m] Double word-by-double word read/write

p : Signal type (X, G, F, Y, D, R)
m : Signal byte address
n : (Used for bit-by-bit read/write only) Signal address bit number (0 to 7)

PMC interface signals are read and written. Bit-by-bit and byte-by-byte read/write operations are possible.
The following signals can be used:

Variable name	Signal type	Read	Write
#IOX #IOXB #IOXW #IOXD	X	Possible	Impossible
#IOG #IOGB #IOGW #IOGD	G	Impossible	Possible
#IOF #IOFB #IOFW #IOFD	F	Possible	Impossible
#IOY #IOYB #IOYW #IOYD	Y	Impossible	Possible
#IOD #IODB #IODW #IODD	D	Possible	Possible
#IOR #IORB #IORW #IORD	R	Possible	Possible

For the valid signal address range, see the specifications of the PMC as well.

When writing to a signal, make the variable unprotected on the PMC signal protection screen (described later) beforehand.

Specify an address by using m and n.
Example:

#IOF[1, 3]	F1.3 bit type
#IOG[1, 5]	G1.5 bit type
#IOFB[32]	F32 byte type
#IOGB[12]	G12 byte type

Read/write operations are performed in the same as for an ordinary macro statement.
Example:

#RV[0]=#IOFB[32]	Assigns F32 to #RV[0].
#IOG[99.3] = 1	Sets G99.3 to 1.

If a signal at a nonexistent address is specified, a PS alarm is issued.

⚠ CAUTION

1 Controls handling other signals, such as a ladder or macro executor, must not write to a signal address being written to by an RTM statement. Ensure that a single control writes to the same byte signal address.
For example, when the G000.0 signal is written to by an RTM statement, do not write to the G000.7 signal from a ladder.
2 Ensure that the same F signal is not read from by an RTM statement and written to by the NC simultaneously.
3 Ensure that the same G signal is not written to by an RTM statement and read from by the NC simultaneously.
4 For word-by-word read/write, specify an even-numbered address and for double word-by-double word read/write, specify an address that is a multiple of four.

NOTE

The system variables described above are not supported for a multi-path PMC.

- PMC signal protection

Whether to enable a write to a signal handled by an RTM statement can be set. This function protects against a malfunction due to incorrect coding.

On the PMC signal protection screen, set whether to enable a write to a signal.
If an RTM statement makes an attempt to write to a signal write-protected on the PMC signal protection screen, a PS alarm is issued at the time of execution.

Set whether to enable a write to each address of Y and G on a byte-by-byte basis.
For each of addresses D and R, set a write-enabled range.

For the unwritable signals (X, F), the screen is not displayed.

Before changing an address on the PMC signal protection range specification screen, perform a clear operation first and then input a new address.

- Input/output

A value set for PMC signal protection can be input/output.

- Input/output format

After outputting PMC signal protection, one file (DIDOENBL.TXT) is created.
Please execute input/output operation in EDIT mode.
The output format is as follows:

L Specification method
  0: Byte specification
  1: Range specification
Q Alphabetic signal address
  0: G, 2: Y, 5: R, 9: D
K (for use in range specification only) Index number (0 to 9)
R Address number for byte specification
  Start address number for range specification
P Protection value for byte specification
  0: Not writable
  1: Writable
  End address number
  for range specification

Example of output

```
%
L0Q2R0000P0
L0Q2R0001P1
:                    } Y0 to Y127 protect information
L0Q2R0127P1
:
L0Q0R0000P1
L0Q0R0001P1
:                    } G0 to G767 protect information
L0Q0R0767P0
L0Q0R1000P1
:                    } G1000 to G1767 protect information
L0Q0R1767P0
:                                                        } Byte specification
L1Q5K0R0002P0014 —— R2 to R14 are writable (Index 0)
:                                                        } Range specification
L1Q9K9R0032P0127 —— D32 to D127 are writable (Index 9)
M02
%
```

17.2.1.2 Real time macro variables (RTM variables)

The real time macro variables (RTM variables) are variables dedicated to real time custom macros.
The RTM variables are classified as volatile real time macro variables (volatile RTM variables) and nonvolatile real time macro variables (nonvolatile RTM variables).

The data of a nonvolatile RTM variable is preserved even when the power is turned off.
The data of a volatile RTM variable is cleared to 0 when the power is turned off.

Format

#RV [m] Volatile RTM variable

m: Volatile RTM variable number (0 to 99)

#RVS [n] Nonvolatile RTM variable

n: Nonvolatile RTM variable number (0 to 31)

NOTE

1 RTM variables can be used with an RTM statement only. RTM variables cannot be used with an NC statement and macro statement.
2 No RTM variable assumes a "null" value.
3 Volatile RTM variables are cleared to 0 by a reset. On the other hand, nonvolatile RTM variables are not cleared to 0 by a reset.

Explanation

- Input/output

RTM variables can be input/output in a specified format.
Both nonvolatile RTM variables and volatile RTM variables can be input/output.

- Input/output format

After outputting RTM variables, one file(RTMMACRO.TXT) is created.
The output format is described below.
The value of an RTM variable is output in hexadecimal as a double-precision floating-point data bit image.
When a nonvolatile RTM variable is output, G10L87 is followed by an RTM variable number and a variable value.
When a volatile RTM variable is output, G10L88 is followed by an RTM variable number and a variable value.

Example of output

```
%
G10L87P0(3FE0000000000000)
G10L87P1(4000000000000000)
:                                  } Nonvolatile RTM variable
G10L87P30(4010000000000000)
G10L87P31(4014000000000000)

G10L88P0(4008000000000000)
G10L88P1(3FD9999999800000)
:                                  } Volatile RTM variable
G10L88P98(3FF0000000000000)
G10L88P99(4010000000000000)
M02
%
```

Execute input/output operation in EDIT mode.

17.2.2 Custom Macro Variables

With real time custom macros, a part of the custom macro variables (part of the system variables) can be handled.

17.2.2.1 System variables

With real time custom macros, position-related information among the system variables of the custom macros can be handled.

- Position information #100001 to #100182 (Attribute: Read only)
Block end position #100001 to #100032
Current position (machine coordinate system) #100051 to #100082
Current position (workpiece coordinate system) #100101 to #100132
Skip position #100151 to #100182

By readying the values of system variables #100001 to #100182, the end position of the previous block, the current positions (machine coordinate system and workpiece coordinate system), and skip signal position can be found.

Variable No.	Position information	Coordinate system
#100001 #100002 : #100032	Block end position on 1st axis Block end position on 2nd axis : Block end position on 32nd axis	Workpiece coordinate system
#100051 #100052 : #100082	Current position on 1st axis Current position on 2nd axis : Current position on 32nd axis	Machine coordinate system
#100101 #100102 : #100132	Current position on 1st axis Current position on 2nd axis : Current position on 32nd axis	Workpiece coordinate system
#100151 #100152 : #100182	Skip position on 1st axis Skip position on 2nd axis : Skip position on 32nd axis	Workpiece coordinate system

NOTE
1 The value of a variable with a number greater than the number of controlled axes is undefined.
2 The end position (ABSIO) of a skip (G31) block is the skip signal ON position if the skip signal is turned on. If the skip signal is not turned on, the block end point is undefined.
3 As block end position information #100001 to #100032, the end positions read in advance are obtained instead of the end positions of the block currently being executed.

- Servo positional deviation #100251 to #100282 (Attribute: Read only)

By reading the values of system variables #100251 to #100282, the servo positional deviation on each axis can be found.

Variable No.	Position information
#100251 #100252 : #100282	Servo positional deviation on 1st axis Servo positional deviation on 2nd axis : Servo positional deviation on 32nd axis

NOTE
The value of a variable with a number greater than the number of controlled axes is undefined.

- Remaining travel distance #100801 to #100832 (Attribute: Read only)

By reading the values of system variables #100801 to #100832, the remaining travel distance on each axis can be read.

Variable No.	Position information
#100801	Travel distance on 1st axis
#100802	Travel distance on 2nd axis
:	:
#100832	Travel distance on 32nd axis

NOTE
1 The value of a variable with a number greater than the number of controlled axes is undefined.
2 System variables not described here are unusable.
3 The name of a system variable cannot be specified.
Do not specify a command as indicated below.
//1 #RV[0]=[#_ABSOT[1]] ;

17.2.2.2 Local variables

The local variables(#1 to #33) cannot be used.

17.3 ARITHMETIC AND LOGICAL OPERATION

With the real time custom macros, the following arithmetic and logical operations can be specified:

Table 17.3 (a) Arithmetic and logical operation

Type of operation	Operation	Description
(1) Definition, substitution	#i=#j	Definition or substitution of a variable
(2) Additive operation	#i=#j+#k	Addition
	#i=#j-#k	Subtraction
	#i=#j OR #k	Logical OR (Bit by bit for 32 bits)
	#i=#j XOR #k	Exclusive OR (Bit by bit for 32 bits)
(3) Multiplicative operation	#i=#j*#k	Multiplication
	#i=#j/#k	Division
	#i=#j AND #k	Logical AND (Bit by bit for 32 bits)
	#i=#j MOD #k	Remainder (#j and #k are rounded to an integer to find a remainder. When #j is negative, #i is also negative.)

Type of operation	Operation	Description
(4) Function	#i=SIN[#j]	Sine (in deg)
	#i=COS[#j]	Cosine (in deg)
	#i=TAN[#j]	Tangent (in deg)
	#i=ASIN[#j]	Arcsine
	#i=ACOS[#j]	Arccosine
	#i=ATAN[#j]	Arctangent (1 argument). ATN is acceptable.
	#i=ATAN[#j]/[#k]	Arctangent (2 arguments). ATN is acceptable.
	#i=ATAN[#j,#k]	Ditto
	#i=SQRT[#j]	Square root. SQR is acceptable.
	#i=ABS[#j]	Absolute value
	#i=BIN[#j]	Binary conversion from BCD
	#i=BCD[#j]	BCD conversion from binary
	#i=ROUND[#j]	Rounding. RND is acceptable.
	#i=FIX[#j]	Discarding fractional digits
	#i=FUP[#j]	Rounding fractional digits upward to an integer
	#i=LN[#j]	Natural logarithm
	#i=EXP[#j]	Exponent with e (2.718……) used as the base
	#i=POW[#j,#k]	Power (#j to #k power)

- Constant specifiable in <expression>

+0.00000000001 to +999999999999
-999999999999 to -0.00000000001
Up to 12 decimal digits can be specified.
If the maximum allowable number of digits is exceeded, PS0004, “INVALID BREAK POINT OF WORDS” alarm is issued.

NOTE
1 The ADP function is not available.
2 With an RTM statement, the external output commands (BPRNT, DPRNT, POPEN, and PCLOS) are unavailable.
3 The FS16i compatibility specifications are not applicable. Bit 0 (F16) of parameter No. 6008 = 1 (with the operation result precision based on the FS16i compatibility specifications) is invalid.
4 For information on decreased precision, see "When bit 0 (F16) of parameter No. 6008 is set to 0" in "Caution concerning decreased precision" in the CUSTOM MACRO chapter.
5 Bit 0 (NAT) of parameter No. 6004 is valid in an RTM command.
6 The setting of bit 5 (SBM) of parameter No. 6000 and bit 7 (SBV) of parameter No. 6000 is invalid. For the single block specifications, see the separate item.

17.4 CONTROL ON REAL TIME MACRO COMMANDS

Explanation

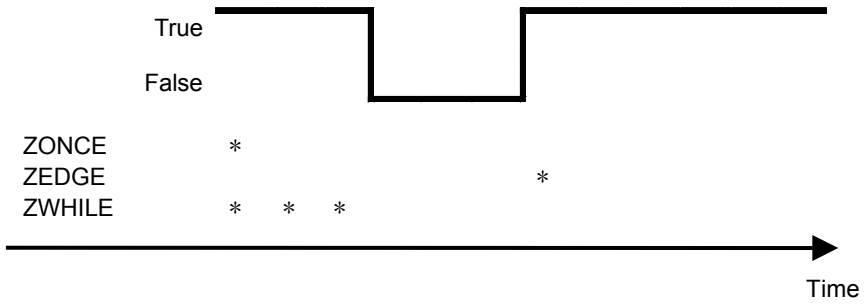
By using a reserved word for controlling statements in an RTM command, the flow of the RTM command can be changed or multiple statements can be controlled as a set of statements. Four reserved words are used to control an RTM command as indicated below.

Reserved word name	Syntax	Meaning
ZONCE	// ZONCE A B	If A is true, B is executed.
ZEDGE	// ZEDGE A B	If A becomes true, B is executed.
ZWHILE	// ZWHILE A B	While A is true, B is executed repeatedly.

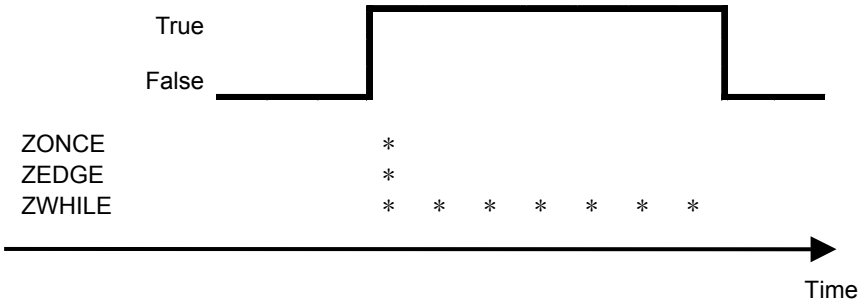
Reserved word name	Syntax	Meaning
ZDO...ZEND	// ZDO B1 B2 B3 ZEND	(Multiple statements) B1, B2, and B3 are sequentially executed.

The timing chart of an RTM command using these reserved words is indicated below. (Multi-statement control ZDO...ZEND is excluded.)
When the condition of a each reserved word is True, it shows '*'.

When condition A makes transitions from True to False to True



When condition A makes transitions from False to True to False



17.4.1 Conditional Branch (ZONCE Statement)

After ZONCE, <conditional-expression> and <real-time-macro-statement> are coded.

- //(n) ZONCE [<conditional-expression>] <real-time-macro-statement>

If <conditional-expression> is true, <real-time-macro-statement> is executed. If <conditional-expression> is false, <real-time-macro-statement> is not executed, but control exits from the ZONCE command to terminate the execution. <conditional-expression> conforms to the custom macro specifications.

// ZONCE A B (If A is true, B is executed.)
If A is true, the command is terminated after executing B.
If A is false, the command is terminated without executing B.

If the workpiece coordinate on the first axis is greater than 30, the Y1.0 signal is output.
// ZONCE [#100101 GT 30.] #IOY[1,0] = 1 ;

If the workpiece coordinate on the first axis is greater than 30 and #RV[0] matches #RV[3], the Y1.0 signal is output.
// ZONCE[[#100101 GT 30.] AND [#RV[0] EQ #RV[3]]] #IOY[1,0]=1 ;

In <real-time-macro-statement>, multiple RTM statements can be coded.
In this case, code the following by using ZDO...ZEND of multi-statement structure:
// ZONCE [<conditional-expression>] ZDO ;
<real-time-macro-statement-1> ;
<real-time-macro-statement-2> ;
:
ZEND ;

If the workpiece coordinate on the second axis is equal to or less than 10, the rapid traverse override value is changed.
// ZONCE [#100102 LE 10.] ZDO ; **#IOG[14,0]=0 ;** **#IOG[14,1]=1 ;** **ZEND ;**

However, if <conditional-expression-1> specifies an axis control command, be sure to use ZDO...ZEND even when a single statement is used.

If the workpiece coordinate on the first axis is greater than 30 and #RV[0] matches #RV[3], a movement on the A-axis starts.
//1 ZONCE [[#100101 GT 30.] AND [#RV[0] EQ #RV[3]]] ZDO ; **G91 G00 A20. ;** **ZEND ;**

Similarly, use ZDO...ZEND for a multi-statement including an axis control command.

If the workpiece coordinate on the second axis is equal to or less than 10, a movement on the V-axis starts and the Y1.0 signal is set to 1.
//1 ZONCE [#100102 LE 10.] ZDO ; **G91 G00 V10. ;** **#IOY[1,0] = 1 ;** **ZEND ;**

17.4.2 Condition Transition (ZEDGE Statement)

After ZEDGE, <conditional-expression> and <real-time-macro-statement> are coded.
// ZEDGE A B (If A becomes true, B is executed.)

- //(n) ZEDGE [<conditional-expression>] <real-time-macro-statement>

When <conditional-expression> makes a transition from false to true, an RTM statement specified after <conditional-expression> is executed. <conditional-expression> conforms to the custom macro specifications.
With the ZEDGE command, the result of the first evaluation of <conditional-expression> is always false.

Explanation

The difference between the ZONCE command and ZEDGE command is that if <conditional-expression> is false, the RTM statement itself is terminated in the case of ZONCE, while the evaluation of <conditional-expression> continues until <conditional-expression> becomes true in the case of ZEDGE. (With a one-shot real time macro command, the evaluation of <conditional-expression> continues until the execution of the NC statement that started operation simultaneously with the command ends.)

On the rising edge of the G address signal 4.3, the workpiece coordinate on the third axis is read.
// ZEDGE [#IOG[4,3] EQ 1] #RV[0]=#100103 ;

In the example above, even if the [#IOG[4,3] EQ 1] is true from the beginning, #RV[0]=#100103 of the RTM statement is not executed. #RV[0]=#100103 is executed when the result of evaluation of [#IOG[4,3] EQ 1] changes from false to true.

As in the case of ZONCE, multiple RTM statements can be coded in <real-time-macro-statement>.
If <conditional-expression-1> specifies an axis control command, be sure to use ZDO...ZEND even when a single statement is used.
Code the following by using ZDO...ZEND of multi-statement structure:
// ZEDGE [<conditional-expression>] ZDO ;
<real-time-macro-statement-1> ;
<real-time-macro-statement-2> ;
:
ZEND ;

On the falling edge of the X address signal, a movement on the B-axis is started and the Y1.0 signal is set to 1.
// ZEDGE [#IOX[1,3] EQ 0] ZDO ; **G91 G00 B10. ;** **#IOY[1,0] = 1 ;** **ZEND ;**

On the rising edge of the G address signal, a movement on the U-axis is started.
// ZEDGE [#IOG[4,3] EQ 1] ZDO ; **G91 G00 U25. ;** **ZEND ;**

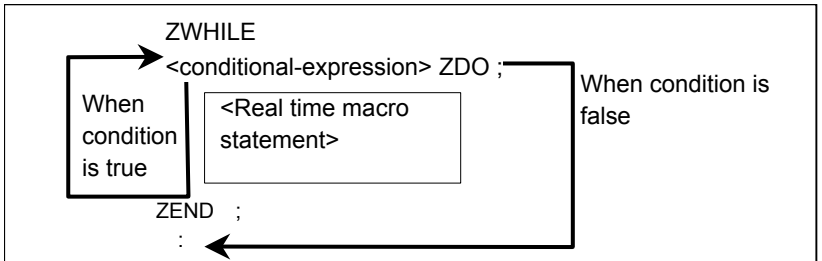
If the remaining travel distance on the second axis is equal to or less than 10, the rapid traverse override value is changed.
// ZEDGE [#100802 LE 10.] ZDO ; **#IOG[14,0]=0 ;** **#IOG[14,1]=1 ;** **ZEND ;**

17.4.3 Repetition (ZWHILE Statement)

After ZWHILE, a <conditional statement> is coded.
// ZWHILE A B (While A is true, B is executed repeatedly.)

- //(n) ZWHILE [<conditional-expression>] <real-time-macro-statement>

While <conditional-expression> is true, the RTM statement specified after <conditional-expression> is executed.
If <conditional-expression> is not satisfied, the ZWHILE statement is terminated, and the next block is processed.



* When the <real time macro statement> is a single statement and is not an axis control statement, ZDO and ZEND are not required.
* When “conditional-expression” is fulfilled and “Real time macro statement” is complete, interpolation cycle time has passed before the next evaluation of "ZWHILE conditional-expression".

Explanation

While <conditional-expression> is true, the command or commands between ZDO and ZEND after ZWHILE are executed.
If <conditional-expression> is not satisfied, the command after ZEND is processed.
The same <conditional-expression> and operators as for the ZONCE statement are used.

While the F address 234.1 signal is 1, an incremental movement on the U-axis is repeatedly performed, and #RV[0] is incremented by 1 each time.
//1 ZWHILE [#IOF[234,1] EQ 1] ZDO ; **G91 G00 U10. ;** ** #RV[0] = #RV[0]+1 ;** **ZEND ;**

17.4.4 Multi-statement (ZDO...ZEND Statement)

- **//(n) ZDO ;**
  <real-time-macro-statement-1> <real-time-macro-statement-2>
  <real-time-macro-statement-3> …
  ZEND ;

One or multiple statements enclosed in ZDO...ZEND are regarded as a one RTM statement (multiple-statement structure).

Explanation

The maximum number of RTM statements in one multi-statement (between ZDO...ZEND) is roughly five, but it is subject to change (Because it depends on the contents of RTM statements).

ZDO...ZEND is used in the following cases:

- When multiple <real time macro statements> are handled as a series of commands

If the workpiece coordinate on the first axis exceeds 30, the rapid traverse override value is changed.
// ZEDGE [#100101 GT 30.] ZDO ; **#IOG[14,0]=1 ;** **#IOG[14,1]=0 ;** **ZEND ;**

- When an axis control command is specified

On the rising edge of the X signal 7.0, a movement on the V-axis is started.
// ZEDGE [#IOX[7,0] EQ 1] ZDO ; **G91 G00 V35. ;** **ZEND ;**

- Nesting

ZONCE, ZEDGE, ZWHILE, and ZDO...ZEND cannot be nested and overlapped.
For details, see the following:

1. ZONCE, ZEDGE, ZWHILE, and ZDO...ZEND may be used any number times.

```
// ZWHILE [···] ZDO ;
    Processing
ZEND ;
  :
// ZONCE[···] ZDO ;
    Processing
ZEND ;
  :
```

2. One ZDO...ZEND range must not overlap another ZDO...ZEND range.

```
// ZONCE [···] ZDO ;
    Processing
// ZWHILE [···] ZDO ;
  :
ZEND ;
    Processing
ZEND ;
```

3. ZONCE, ZEDGE, ZWHILE, and ZDO...ZEND must not be nested.

```
// ZWHILE [···] ZDO ;
      :
  ZONCE [···] ZDO ;
        :
    ZEDGE [···] ZDO ;
      Processing
    ZEND ;
      :
  ZEND ;
    :
ZEND ;
```

- Endless loop

An endless loop is formed if the conditional expression enclosed in brackets after the ZWHILE statement is always satisfied.

Example)

In the example below, #RV[0] is counted up unconditionally.

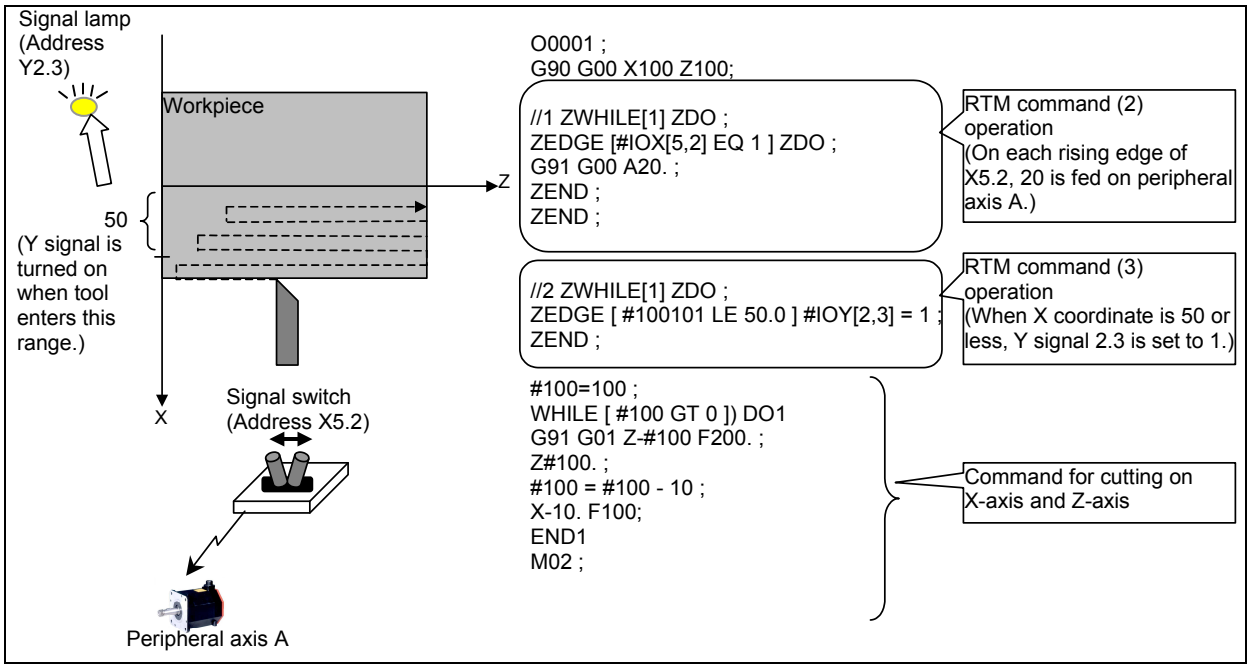
```
// ZWHILE [1] #RV[1]=#RV[1]+1 ;
```

Sample program

The sample program below exercises the following three control operations at the same time.

(1) A cutting operation is performed on the X-axis and Z-axis.
(2) On each rising edge of the X signal 5.2, 20 is fed on the peripheral axis A.
(3) When the workpiece coordinate on the X-axis (#100101) is equal to or less than 50, the Y signal 2.3 is set to 1.

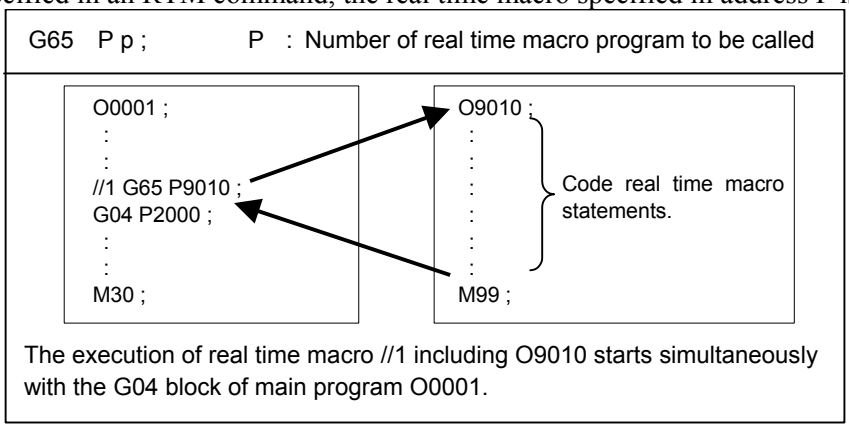
The operation of (1) is coded in the main NC statement.
The operation of (2) is coded in the first modal RTM command (//1).
The operation of (3) is coded in the second modal RTM command (//2).



17.5 MACRO CALL

A series of RTM statements can be formed into a subprogram, which can be called from the main program.

When G65 is specified in an RTM command, the real time macro specified in address P is called.



Explanation

- Calling

In address P after G65, specify the program number of a real time custom macro to be called.

NOTE

1 In argument P of G65, only a constant value can be used. No value can be specified through a variable.
Example:
// G65 P9010 ; Correct
// G65 P#RV[0] ; Incorrect

2 The inclusion of another NC command (such as G01 X100.0 G65 Pp) is not allowed. If another NC command is included, alarm PS0127, “DUPLICATE NC,MACRO STATEMENT” is issued.

NOTE
3 The G65 block for calling a real time macro does not make a single block stop.
4 On the other hand, a real time macro program called by real time macro calling makes a single block stop.

- Call destination real time program

In a called real time macro program, only an RTM statement can be coded.

In a called real time macro program, no additional RTM command may be executed. (The RTM command symbol '//' may not be coded.) For example, do not execute the following program:

```
O0001 ;                    O9010 ;
// G65 P9010 ;             // #RV[0]=1 ;
M02 ;                      M99 ;
```

Code M99 in the last block only.
For example, do not execute the following program:

```
O0001 ;                    O9010 ;
// G65 P9010 ;             ZEDGE [#RV[0] EQ 1] M99 ;
M02 ;                      G91 G00 X50. ;
                           M99 ;
```

When executing an RTM statement in a subprogram, code an NC statement before the M99 block for returning to the calling program.

```
O0001 ;                    O9010 ;
G65 P9010 ;                // #RV[0] = 20 ; Real time macro statement
M02 ;                      G04 ;           NC statement
                           M99 ;
```

- Format

Specify G65 at the start of a block.

- Call nesting level

Macro call nesting is not allowed.

- Differences from macro call using a custom macro

Macro call using a real time custom macro and macro call using a custom macro differs as described below.

- In macro call using a custom macro, an argument (data to be passed to a macro) and the number of repetitions can be specified. In macro call using a real time custom macro, such information cannot be specified.
- With a real time custom macro, other types of macro call (macro call using G66, G66.1, G, and M code) and subprogram call are not allowed.

17.6 OTHERS

If an axis control command is followed by a macro command in an RTM command, the execution of the macro command starts when the axis control command is completed or deceleration starts.
If deceleration on the X-axis starts upon completion of distribution according to the axis control command of (1), for example, the macro command of (2) is executed. If acceleration/deceleration is not applied on the X-axis, (2) is executed upon completion of distribution on the X-axis.

```
// ZDO ;
G91 G00 X30 ;          (1) Axis control command of the RTM statement
#RV[0] = 1 ;           (2) Macro command of the RTM statement
ZEND ;
```

If an RTM command is executed in a custom macro WHILE DO to END loop, and the same RTM is looked ahead while the RTM command is being executed, a PS alarm may be issued.
For example, if the command below is executed, a PS alarm is issued.

```
WHILE [1] DO1 ;
// #RV[0] = #RV[0] + 1 ;
#100=#100+1 ;
END1 ;
```

In this example, insert G04 ; block between a #100=#100+1; and END1;.

17.7 AXIS CONTROL COMMAND

In an RTM statement, a G code for specifying a movement can be specified. For axis control, the PMC axis control interface is used. The specifications differ from the specifications for the G code used with an NC statement.

Format

```
// ZDO ;
G_ IP _ F_ ;
  :
ZEND ;
```

For an axis control command, ZDO...ZEND of multi-statement structure is used for coding even if a single statement (single RTM statement) is used.

⚠ CAUTION
Do not specify an NC statement for an axis to be controlled with an RTM statement. Moreover, do not exercise axis control with an RTM statement on an axis to be controlled with an NC statement.

NOTE
1 In one block, only one axis may be specified. Do not specify two or more axes in the same block.
2 In the 3-dimensional coordinate conversion mode, the polygon mode, etc., an RTM statement concerning that axis cannot be specified.

Explanation

- Setting

For an axis control command in an RTM statement, the interface of a PMC axis control group set by a parameter beforehand is used.
For an axis to be controlled with an RTM statement, set a group to be used with parameter No. 8010. Then, set bit 0 (XRT) of parameter No. 8011 for the axis to 1.

NOTE
An axis for which bit 0 (XRT) of parameter No. 8011 is set to 1 is dedicated to real time custom macros, so that such an axis cannot be used with PMC axis control.

- Relationship with PMC axis control

Axis control based on an RTM statement uses the PMC axis control interface. So, the specifications for a move command in each block within an RTM statement are generally equivalent to the specifications for the PMC axis control command.
For the detailed specifications and restrictions related to axis control, refer to the specifications of PMC axis control as well.

NOTE
1 An axis used with an RTM statement must not be specified from PMC axis control.
2 A PMC axis control group used with an RTM statement must not be specified from PMC axis control.

- Operation command code

The table below indicates the G codes that can be specified in an RTM statement.
The ◤ mark in the table indicates modal G codes when an RTM command starts.

Code	Group	Meaning
◤ G00	01	Positioning
G01		Feed at specified feedrate
G04	00	Dwell
G28		Reference position return
G53		Machine coordinate system selection
G90	03	Absolute command NOTE 1
◤ G91		Incremental command NOTE 2
◤ G94	05	Feed per minute
G95		Feed per revolution

NOTE
1 The absolute command (G90) is valid for machine coordinate system selection (G53) only.
2 The incremental command (G91) is valid for positioning (G00), linear interpolation (G01), and reference position return (G28) only.
3 In T series G code system A as well, not G98/G99 but G90/G91 are valid with an RTM command.

- Modal information

Basically, modal information such as F and modal G codes in an RTM statement is independent in an NC statement and each RTM statement.
Modal information in an RTM statement is initialized when the execution of the RTM statement is started (when a block specified with // is executed).
The initial value of modal information in an RTM statement is as follows:

State of the G codes marked with ◤ in the operation command code list
F code : F0

⚠ CAUTION
With the G codes (inch input/metric input) of group 06, the same information as the modal information of an NC statement is used in an RTM statement. Do not change the modal information of group 06 with an NC statement in a block after the first RTM statement coded in a program.

NOTE
With an RTM command, bit 0 (G01), bit 3 (G91), and bit 4 (FPM) of parameter No. 3402 are invalid when the power is turned on or in the cleared state.

Example 1)

Modal information can be controlled independently in an NC statement and each RTM statement.

```
O0001 ;
G90 G01 X100. Y100. F500. ;        (1)
//1 ZDO ;
Z50. ;                             (2)
G01 Z100. F100. ;                  (3)
ZEND ;
//2 ZDO ;
G01 A30. F200. ;                   (4)
A50. ;                             (5)
ZEND ;
X200. ;                            (6)
:
```

The modal information of command (2) is G00, G91, and G94, regardless of the modal information (G90 G01 set in (1)) of the NC statement.
The modal information of command (5) is G01, G91, G94, and F200.. This modal information is not affected by the modal information (command (1)) of the NC statement and the modal information (commands (2) and (3)) in the RTM statement with modal ID=1.
The modal information of command (6) is not affected by the modal information (commands (2) to (5)) in the RTM statement, so that the modal information is G01, G90, and F500..

Example 2)

Modal information is initialized when the execution of each RTM command is started. Even if the same program includes RTM commands with the same ID, the modal information of the RTM command executed first is not inherited by the RTM command executed next.

```
O0001 ;
G90 G01 X100. Y100. F500. ;
//1 ZDO ;
G01 Z100. F3000. ;                 (1)
ZEND ;
X200. ;
//1 ZDO ;
Z200. ;                            (2)
ZEND ;
X300. ;
:
```

The modal information of command (2) is G91 and G00, regardless of command (1).

- Single block stop

If an NC statement is placed in the single block stop state, for example, by the single block stop signal SBK, the axis control command of an RTM statement also comes to a single block stop.
However, the automatic operation start signal STL is turned off when all conditions on the NC statement side are satisfied, even if the axis control command of an RTM statement is not terminated.

If an RTM command consists of multiple statements and an axis control command is coded in multiple blocks, only the block of the RTM statement that is currently executing an axis command can be brought to a single block stop by setting the group-by-group single block stop signal (ESBKg) for PMC axis control corresponding to the axis to 1.

- Feed hold

Even when an NC statement enters the automatic operation halt state, the axis control command of an RTM statement does not stop immediately but stops at the time of termination of the block currently being executed. To stop also the axis control command of an RTM statement immediately when the NC statement enters the automatic operation halt state, control the temporary stop signal (ESTPg) for PMC axis control of the related group by monitoring the automatic operation halt in-progress signal (SPL).

- Reset

Even when the CNC is reset by an MDI reset, the external reset signal (ERS), or the reset and rewind signal (RRW), the axis control command of an RTM statement does not stop immediately but stops at the time of termination of the block currently being executed. When the emergency stop state is set, however, the RTM statement also stops immediately.
During execution of the axis control command of an RTM statement, the real time macro being executed can be stopped independently of the NC statement by setting the group-by-group reset signal (ECLRg) for PMC axis control corresponding to the axis to 1. To stop also the axis control command of an RTM statement when the CNC is reset, control the reset signal (ECLRg) for PMC axis control of the related group by monitoring the reset signal (RST).

- Alarm stop

Even when a PS alarm is issued with an NC statement, the axis control command of the RTM statement being executed does not stop immediately but stops at the time of termination of the block currently being executed. Moreover, even when an overtravel alarm is issued for an axis other than the axis controlled by the RTM statement being executed, the RTM statement being executed does not stop immediately but stops at the time of termination of the block currently being executed.

- Alarm reset

When the group-by-group alarm signal (EIALg) for PMC axis control corresponding to the axis specified by an RTM statement is set to 1, correct the cause of the alarm, then reset the CNC, and reset PMC axis control with the reset signal (ECLRg) for PMC axis control of the related group.
(Tip) In the following cases, the alarm signal (EIALg) is set to 1:

(1) When a servo alarm is issued
(2) When an overtravel alarm is issued
(3) When P/S0130 alarm is issued
(4) When P/S0139 alarm is issued

- Interlock

In an RTM statement, the interlock signal (*IT) for an NC statement, interlock signal for each axis (*ITn), interlock signal for each axis direction (+ITn/-ITn) are invalid. Instead, the axis control temporary stop signal (ESTPg) for PMC axis control is valid in an RTM statement. When temporarily stopping the axis controlled by an RTM statement, control the corresponding axis control temporary stop signal (ESTPg) for PMC axis control.

The cutting block start interlock signal *CSL (G8.1) and the block start interlock signal *BSL (G8.3) are invalid.

- Machine lock

The same machine lock signals (all axes/each axis) as used with an NC statement are used. However, by disabling machine lock for PMC axis control with the following parameters, machine lock can be disabled for the axis being controlled by an RTM statement:

Bit 0 (MLE) of parameter No. 8001
Bit 1 (MLS) of parameter No. 8006

- Dry run

With bit 2 (OVE) of parameter No. 8001, whether to use the dry run signal (DRN) for an NC statement or the dry run signal (EDRN) for a PMC axis can be chosen. When enabling a dry run feedrate for rapid traverse in an RTM statement, set bit 3 (RDE) of parameter No. 8001 to 1. If a feed command specifying a feedrate is specified when the manual rapid traverse selection signal (RT or ERT) is set to 1, the cutting feedrate is used as the feedrate during dry run. ("Dry run feedrate × maximum manual feedrate override value" is not applicable.)

- In-position check

In the in-position state, the in-position signal (EINPg) is set to 1.
When bit 6 (NCI) of parameter No. 8004 is set to 1, no in-position check is made during axis control based on an RTM statement. The setting of bit 5 (NCI) of parameter No. 1601 for an NC statement is invalid. (However, the in-position signal (INPx) for an NC statement is affected by the value of NCI.)

- Manual absolute

The manual absolute signal (*ABSM) is invalid.

- External deceleration

By setting bit 0 (EDC) of parameter No. 8005 to 1, the external deceleration function can be used also for the axis being controlled by an RTM statement. Note, however, that the external deceleration function is unusable for feed with a specified feedrate (feed per revolution). For an external deceleration rate, the external deceleration rate setting for rapid traverse (parameter No. 1427, No. 1441, and No. 1444) is valid, regardless of the type of feed.
The external deceleration function for the axis being controlled by an RTM statement is also valid for feed with a specified feedrate (feed per minute), regardless of the settings of bit 4 (EDP) of parameter No. 1005 and bit 5 (EDM) of parameter No. 1005. (The enabling of rapid traverse only is impossible.)

- Mirror image

At enabling mirror image for the axis being controlled by an RTM statement, set bit 0 (EMR) of parameter No. 8008 to 1, and set either bit 0 (MIRx) of parameter No.0012 or mirror image signal MIx to '1'.
Programmable mirror image is not available.

NOTE
1 The same group as the PMC axis control group used for axis control cannot be executed with an RTM statement.
2 No NC statement can specify an axis being controlled by an RTM statement.

NOTE

3 An alarm is issued if, during execution of an RTM statement, an attempt is made to execute another RTM statement with the same ID. In the program below, for example, the RTM statement of (1) operates using the NC statement of (2) as a trigger, and the RTM statement of (3) operates using the NC statement of (4) as a trigger. An alarm is issued if (1) is still operating when (4) is started after completion of the interpolation of (2).

```
//1 ZDO ;
G91 G00 A100. ;        (1)    RTM statement
ZEND ;
G01 X30. Y20. ;        (2)    NC statement
//1 ZDO ;
G91 G00 B100. ;        (3)    RTM statement
ZEND ;
X40. Y50. ;            (4)    NC statement
```

4 When T series G code system A is used, address words (such as U, W, H, and V) for incremental commands must not be coded. Otherwise, alarm PS0413 is issued.

5 An RTM statement for performing axis control proceeds to the next command at the end of distribution. For example, in the program below, #RV[0] counts up at the time X50. distribution ends and deceleration starts.

```
// ZDO ;
G00 X50. ;
#RV[0]=#RV[0]+1 ;
ZEND ;
```

Command details

- Rapid traverse

A movement is made at a rapid traverse rate on an axis from the current position to the point separated by a specified value.

Format

```
// ZDO ;
G91 G00 IP _ ;
ZEND ;
```

G91 : G code for incremental command
IP _ : Travel distance

NOTE

1 Only one axis can be specified in one block.
2 The absolute command (G90) cannot be specified.
3 The rapid traverse overlap function cannot be used.
4 When using this function, be sure to set bit 0 (RPD) of parameter No. 8002 to 0.

- Rapid traverse override
  With bit 2 (OVE) of parameter No. 8001, whether to use the rapid traverse override signal (ROV) for an NC statement or the rapid traverse override signal (EROV) dedicated to PMC axis control can be chosen.

NOTE
Even if bit 4 (RF0) of parameter No. 1401 is set to 1, rapid traverse does not stop with a cutting feed override of 0%.

- Feed with a specified feedrate (feed per minute)

A movement is made at a feedrate specified in F on an axis from the current position to the point separated by a specified value.

Format

```
// ZDO ;
G94 G91 G01 IP _ F_ ;
ZEND ;
```

G94 : G code for feed per minute
G91 : G code for incremental command
IP __: Travel distance
F__ : Feedrate command (mm/min or inch/min)

NOTE
1 Only one axis can be specified in one block.
2 The absolute command (G90) cannot be specified.
3 The block overlap function cannot be used.
4 When IS-A is used, a feedrate below 10 mm/min is discarded.
5 The feedrate cannot be clamped to a parameter-set maximum cutting feedrate.
6 No time is added to the cumulative cutting time.
7 Even if this command is executed, the cutting in-progress signal is not output.
8 The operation between blocks for feed with a specified feedrate is performed in the cutting mode.
9 For this command, the feedrate arrival signal SAR <G029.4> cannot be used.

- Feedrate specification range
  The table below indicates the feedrate specification range.

	Linear axis		Rotation axis (deg/min)
	Metric input (mm/min)	Inch input (inch/min)	
IS-A	10. to 240000.	0.1 to 6553.5	10. to 240000.
IS-B	1. to 65535.	0.01 to 655.35	1. to 65535.
IS-C	0.1 to 6553.5	0.001 to 65.535	0.1 to 6553.5
IS-D	0.01 to 655.35	0.0001 to 6.5535	0.01 to 655.35
IS-E	0.001 to 65.535	0.00001 to 0.65535	0.001 to 65.535

NOTE
1 Be sure to set the following parameters to 0:
Bit 3 (F10) of parameter No. 8002
Bit 4 (EFD) of parameter No. 8006
Bit 4 (PF1) of parameter No. 8002
Bit 5 (PF2) of parameter No. 8002
When a value other than 0 is set, the feedrate specification range changes, depending on the set parameter.
2 A value less than the lowest feedrates indicated above is discarded.

- Feedrate override
  With bit 2 (OVE) of parameter No. 8001, whether to use the feedrate override signal (*FV) for an NC statement or the feedrate override signal (*EFOV) dedicated to PMC axis control can be chosen.

NOTE
1 The second feedrate override function cannot be used.
2 The feedrate override function cannot be disabled using #3004.

- Override cancel
  With bit 2 (OVE) of parameter No. 8001, whether to use the feedrate override cancel signal (OVC) for an NC statement or the feedrate override cancel signal (EOVC) dedicated to PMC axis control can be chosen.

- Operation between blocks
  If feed with a specified feedrate is performed in succession in RTM statements, no stop occurs between blocks, but the next block is executed. With the accumulated pulse zero check signal (ELCKZg) for PMC axis control, an accumulated pulse zero check can be made between blocks specifying feed with a specified feedrate.

NOTE
1 The error detect signal (SMZ) cannot be used.
2 With the accumulated pulse zero check signal (ELCKZg) for PMC axis control, an accumulated pulse zero check can be made between blocks specifying feed with a specified feedrate for feed on the same axis only.

- Acceleration/deceleration time constant
  For an acceleration/deceleration time constant to be used for feed with a specified feedrate in an RTM statement when exponential acceleration/deceleration is used, whether to use the time constant for an NC statement or the time constant dedicated to PMC axis control can be chosen using parameter No. 8030.

NOTE
Look-ahead acceleration/deceleration before interpolation cannot be used.

- Feed with a specified feedrate (feed per revolution)

A movement is made at a feedrate specified in F on an axis from the current position to the point separated by a specified value.

Format

```
// ZDO ;
G95 G91 G01 IP _ F_ ;
ZEND ;
```

G95 : G code for feed per revolution
G91 : G code for incremental command
IP _ : Travel distance
F_ : Feedrate command (mm/rev or inch/rev)

NOTE
1 Only one axis can be specified in one block.
2 The absolute command (G90) cannot be specified.
3 The block overlap function cannot be used.
4 Be sure to set the parameters below to 0. If a value other than 0 is set, the feedrate specification range changes, depending on the set parameter.
  Bit 6 (FR1) of parameter No. 8002
  Bit 7 (FR2) of parameter No. 8002
5 The feedrate is clamped to the maximum cutting feedrate set in parameter No. 8022. (Parameter No. 1430 for an NC statement is not valid for this command.)
6 To use feed per revolution, a position coder is required at all times. (Bit 0 (NPC) of parameter No. 1402 is not applicable to this command.)
7 No time is added to the cumulative cutting time.
8 Even if this command is executed, the cutting in-progress signal is not output.
9 The operation between blocks for feed with a specified feedrate is performed in the cutting mode at all times.
10 For this command, the feedrate arrival signal SAR <G029.4> cannot be used.

- Feedrate specification range
  The table below indicates the feedrate specification range.

	Linear axis		Rotation axis (deg/rev)
	Metric input (mm/rev)	Inch input (inch/rev)	
T series	0.001 to 65.535	0.000001 to 0.65535	0.001 to 65.535
M series	0.01 to 500.00	0.0001 to 6.5535	0.01 to 500.00

- Feedrate override
  With bit 2 (OVE) of parameter No. 8001, whether to use the feedrate override signal (*FV) for an NC statement or the feedrate override signal (*EFV) dedicated to PMC axis control can be chosen.

NOTE
1 The second feedrate override function cannot be used.
2 The feedrate override function cannot be disabled using #3004.

- Override cancel
  With bit 2 (OVE) of parameter No. 8001, whether to use the feedrate override cancel signal (OVC) for an NC statement or the feedrate override cancel signal (EOVC) dedicated to PMC axis control can be chosen.

- Operation between blocks
  If feed with a specified feedrate is performed in succession in RTM statements, no stop occurs between blocks, but the next block is executed. With the accumulated pulse zero check signal (ELCKZg) for PMC axis control, an accumulated pulse zero check can be made between blocks specifying feed with a specified feedrate.

NOTE
1 The error detect signal (SMZ) cannot be used.
2 With the accumulated pulse zero check signal (ELCKZg) for PMC axis control, an accumulated pulse zero check can be made between blocks specifying feed with a specified feedrate for feed on the same axis only.

- Acceleration/deceleration time constant
  For an acceleration/deceleration time constant to be used for feed with a specified feedrate in an RTM statement when exponential acceleration/deceleration is used, whether to use the time constant for an NC statement or the time constant dedicated to PMC axis control can be chosen using parameter No. 8030.

NOTE
Look-ahead acceleration/deceleration before interpolation cannot be used.

- Reference position return

A movement is made at the rapid traverse rate to the first reference position on a specified axis. Upon completion of reference position return, the return completion lamp is turned on.

Format

```
// ZDO ;
G91 G28 IP 0 ;
ZEND ;
```

G91 : G code for incremental command
IP 0 : Specify an axis on which reference position return is to be performed.
Be sure to specify 0 as the travel distance.

NOTE
1 Only one axis can be specified in one block.
2 No intermediate point may be specified. (Be sure to specify 0 as the travel distance. Otherwise, alarm PS0410 is issued.) If a movement needs to be once made to an intermediate point, specify positioning with rapid traverse (G00)/machine coordinate system selection (G53) then specify this command.
3 The absolute command (G90) cannot be specified.
4 When using this function, be sure to set bit 0 (RPD) of parameter No. 8002 to 0.
5 Before this command can be specified, a reference position must be established. (With this command, do not specify an axis on which a reference position is not established.)
6 Before specifying this command with the T series, cancel tool offset and tool-nose radius compensation. Before specifying this command with the M series, cancel cutter compensation, tool length compensation, and tool offset.

- Rapid traverse override
  With bit 2 (OVE) of parameter No. 8001, whether to use the rapid traverse override signal for an NC statement or the rapid traverse override signal dedicated to PMC axis control can be chosen.

NOTE
Even if bit 4 (RF0) of parameter No. 1401 is set to 1, rapid traverse does not stop with a cutting feed override of 0%.

- Machine coordinate system selection

When a position in the machine coordinate system is specified, a movement is made to the position on the axis by rapid traverse. The G53 code for machine coordinate system selection is a one-shot G code, so that a command in the machine coordinate system is valid in the block that specifies G53. When moving the tool to a machine-specific position such as a tool change position, perform programming in the machine coordinate system based on G53.

Format

```
// ZDO ;
G90 G53 IP _ ;
ZEND ;
```
G90 : G code for absolute command
IP _ : Position in machine coordinate system

NOTE
1 Only one axis can be specified in one block.
2 The incremental command (G91) cannot be specified.
3 When using this function, be sure to set bit 0 (RPD) of parameter No. 8002 to 0.
4 Before this command can be specified, a reference position must be established. (With this command, do not specify an axis on which a reference position is not established.)
5 Before specifying this command with the T series, cancel tool offset and tool-nose radius compensation. Before specifying this command with the M series, cancel cutter compensation, tool length compensation, and tool offset.

- Rapid traverse override
  With bit 2 (OVE) of parameter No. 8001, whether to use the rapid traverse override signal for an NC statement or the rapid traverse override signal dedicated to PMC axis control can be chosen.

NOTE
Even if bit 4 (RF0) of parameter No. 1401 is set to 1, rapid traverse does not stop with a cutting feed override of 0%.

- Dwell

By specifying a dwell, the execution of the next block is delayed by the specified time. (Dwell per second)
Dwell per second is executed irrespective of bit 1 (DWL) of parameter No.3405.

Format

```
// ZDO ;
G04 IP _ ;
ZEND ;
```
IP _ : Axis address where bit 0 (XRT) of parameter No. 8011 was set

- Dwell specification range
  The table below indicates the dwell specification range.

Increment system	Command value range	Dwell time unit
IS-A	0.01 to 999999.99	s
IS-B	0.001 to 99999.999	
IS-C	0.0001 to 9999.9999	
IS-D	0.00001 to 999.99999	
IS-E	0.000001 to 99.999999	

When the diameter is specified, whether to specify a radius value or diameter value for dwell operation can be specified using bit 1 (CDI) of parameter No.8005.

NOTE
1 Address P cannot be specified. Only the axis address where bit 0 (XRT) of parameter No.8011 was set to 1 can be specified.
2 When specified without the decimal point, the specification unit depends on inch/metric.
3 When the increment system is IS-C and bit 1 (DWE) of parameter No.8002=1, the command value becomes 1/100.

17.8 NOTES

- Address without the decimal point

In general, an NC address without the decimal point is subject to calculator-type decimal point input when bit 0 (DPI) of parameter No. 3401 or bit 0 (AXDx) of parameter No. 3455 is set to 1. In other cases, an NC address without the decimal point is regarded as a least input increment.

- Indirect axis address specification

When indirect axis address specification is used, use a direct reference argument. No indirect reference may be used.
Example)

AX[1]	Correct
AX[#RV[0]]	Incorrect

- Diameter specification/radius specification

For an axis to be controlled with an RTM statement, select radius specification. (Set bit 3 (DIA) of parameter No. 1006 to 0.)

- Alarm PS0224

If bit 0 (ZRN) of parameter No. 1005 is set to 0, and an axis control command is specified in an NC statement before a reference position return operation is not performed even once after power-up, alarm PS0224, “ZERO RETURN NOT FINISHED.” is issued. In axis control using an RTM statement, the alarm is not issued even when bit 0 (ZRN) of parameter No. 1005 is set to 0, unless bit 6 (EZR) of parameter No. 8006 is set to 1.

- Axis removal

Even when axes are removed, interlock is not applied to the axis being controlled by an RTM statement.

- Stroke limit check before movement

No stroke limit check before movement is made with a block in an RTM statement.

- Others

For the axis being controlled by an RTM statement, the functions disabled for PMC axis control cannot be used.

- Other disabled functions

For the axis being controlled by an RTM statement, the functions listed below as examples are disabled. Do not specify these functions for the axis being controlled by an RTM statement.

- Auxiliary function
- Feed-forward
- Advanced preview feed-forward
- Fine acceleration/deceleration
- Automatic corner deceleration
- One-digit F code feed

- Scaling
- Coordinate system rotation
- Polar coordinate interpolation
- Balance cutting
- Feed stop
- Constant surface speed control
- Positioning function based on optimal acceleration, etc.

⚠ CAUTION
In an RTM statement, do not specify an axis used with any of the following functions:
1 Axis synchronous control
Be sure to set the axis synchronous control selection signal (SYNCn) and axis synchronous control manual feed selection signal (SYNCJn) of the axis to 0.
2 Synchronous/composite control, superimposed control
3 Index table indexing
4 3-dimensional coordinate system conversion
5 Rigid tapping
6 Compound machining
7 Polygon turning
8 Cs contour control
9 AI contour control I/II

17.9 LIMITATION

Major general notes on RTM commands are provided below.

- Background drawing

The RTM command has no effect in background drawing. Do not specify an RTM command during background drawing.

- Interrupt-type custom macro

In an interrupt-type custom macro, no RTM command can be coded.

- Macro executor

In a macro executor, no RTM command can be coded.
Moreover, no macro executor can be coded from an RTM command.
In a series of programs, do not code a macro executor (execution macro) and RTM command at the same time.
In addition, observe the following:

- Do not code an RTM command in a program that uses an execution macro.
- Do not call an execution macro from a program with an RTM command coded.
- Do not code an RTM command in an execution macro.
- Do not use an execution macro as an RTM command trigger.

- Sequence number

No sequence number can be coded in an RTM command. No RTM command can be searched for a sequence number.

- Optional block skip

Optional block skip is unusable.
A slash (/) that appears in the middle of <expression> (enclosed in [ ] on the right-hand side of an arithmetic/logical expression) is regarded as a division operator; it is not regarded as an optional block skip code.

- Single block

When an RTM command is used, a single block stop generally occurs according to an NC statement, regardless of the values of bit 5 (SBM) of parameter No. 6000 and bit 7 (SBV) of parameter No. 6000. However, no single block stop occurs according to G65 for real time macro call.

- Processing

If there are many RTM statements between NC statements, the command flow may break.

- DNC operation

DNC operation is disabled.

- Operation in each event

If an event such as an emergency stop or alarm occurs during execution of an RTM command, the NC command and RTM command generally operate as indicated below.

Event	NC command	RTM command consisting of a macro command	RTM command including an axis control command
PS alarm issued in NC command execution	Operation stops when the block involving the PS alarm cause starts.	The RTM command stops when the NC command stops.	If the RTM statement being executed when the NC command stops is an axis control command, operation on the axis stops when the block stops.
Alarm issued in RTM command execution	Operation stops with the alarm.	The RTM statement stops at that point. Other RTM commands stop when the NC command stops.	The RTM statement stops at that point. Other RTM commands stop operation on the axis when axis control by the RTM statement being executed when the NC statement stops ends.
Single block (SBK signal)	Operation stops when the command being executed ends.	The RTM command is suspended when the NC command stops. When the NC command restarts, the RTM command restarts.	If the RTM statement being executed when the NC command stops is an axis control statement, operation on the axis stops when the block ends.
Feed hold (*SP signal)	A gradual stop occurs.	The RTM command stops when the NC command stops. When the NC command restarts, the RTM command restarts.	If the RTM statement being executed when the NC command stops is an axis control statement, operation on the axis stops when the block ends.
NC command stopped by a reset (ERS signal)	A gradual stop occurs, and the command also ends.	The RTM command stops when the NC command stops.	If the RTM statement being executed when the NC command stops is an axis control command, operation on the axis stops when the block ends.
Emergency stop	An immediate stop occurs, and the command also ends.	The command immediately ends.	Operation on the axis immediately stops, and the command also immediately ends.

Event	NC command	RTM command consisting of a macro command	RTM command including an axis control command
End of the NC command	-	If the RTM command being executed is a one-shot RTM command, the command ends. If the RTM command being executed is a modal RTM command, the command continues execution.	If the one-shot RTM command being executed when the NC command ends is an axis control command, operation on the axis stops when the block ends.
Machine lock (MLK signal)	The NC command is valid.	The RTM command operates.	The RTM command can be enabled. For details, see the description of the axis control command.
Dry run (DRN signal)	The NC command is valid.	The RTM command operates.	The RTM command can be enabled. For details, see the description of the axis control command.
Interlock (*IT signal)	The NC command is valid.	The RTM command operates.	The RTM command can be enabled. For details, see the description of the axis control command.

18 PROGRAMMABLE PARAMETER INPUT (G10)

Overview

The values of parameters and pitch error compensation data can be entered in a program. This function is used for setting pitch error compensation data when attachments are changed or the maximum cutting feedrate or cutting time constants are changed to meet changing machining conditions.
The input format of pitch error compensation data is changed depending on bit 0 (APE) of parameter No. 3602.

Format

- Parameter entry mode

```
G10L52 ;          Parameter entry mode setting
N_(Q_)R_ ;        For parameters other than the axis type or spindle type
N_P_(Q_)R_ ;      For axis type or spindle type parameters
  :
  :
G11 ;             Parameter entry mode cancel
```

N_ : Parameter number
R_ : Parameter setting value (Leading zeros can be omitted.)
(Q_) : Bit number 0 to 7 (to be set when a bit type parameter is input)
(Enabled when bit 4 (G1B) of parameter No.3454 = 1.)
P_ : Axis number 1 to maximum controlled axis number (to be specified when an axis type parameter or spindle type parameter is specified)

NOTE
G10L52 cannot be used to enter pitch error compensation data.

- Pitch error compensation data entry mode

```
G10L50 ;          Pitch error compensation data entry mode setting
N_(L1)R_ ;        Pitch error compensation data entry
  :
G11 ;             Pitch error compensation data entry mode cancel
```

N_ : Compensation position number for pitch errors compensation +10,000
L1 : Input format of pitch error compensation data (total value type)
R_ : Pitch error compensation data

NOTE
G10L50 cannot be used to enter parameter.

Explanation

- Parameter number(N_)

Programmable parameter input (G10L52) enables writing to a parameter for a specific path by adding that path number to the parameter number to be specified (N_). The format for the parameter number is as follows:

Nppxxxxxxx : Add a path number to the high-order 8th and 9th digits of a parameter number.
For pp, set a path number, and for xxxxxxx, set a parameter number. If a path number is omitted or if 0 is set, writing to a parameter for the local path is assumed, as usual.

NOTE
1 If a path number out of the range is specified, alarm PS0115 is issued.
2 To read and write data for the 10th path, set bit 0 (F16) of parameter No. 6008 to 0.
If parameter bit F16 is 1, the calculation precision is up to the 8th digit. Thus, if a parameter number is determined with a calculation command such as #100= PRM[100000000 +#1] when the data for the 10th path is read or written, the correct number may not be determined.
3 Pitch error compensation data entry does not enable selection of a path.

- Setting value (R_)

Do not use a decimal point in the setting (R_) of a parameter or pitch error compensation data. To change a bit type parameter if bit 4 (G1B) of parameter No. 3454 is 1, specify 0 or 1. If a value other than 0 and 1 is specified, alarm PS1144, "G10 FORMAT ERROR", is issued.
As the value of R, a custom macro variable can be used.
When a parameter of real type is used, set an integer value in (R_) according to the increment system of the parameter.

- Bit number (Q__)

Bit number (Q_) is effective if bit 4 (G1B) of parameter No. 3454 is 1. To set a bit type parameter, set a number in the range of 0 to 7.
A custom macro variable can be used as the value of Q.

- Axis number (P_)

As the axis number (P_), specify the order of a controlled axis to be displayed on the CNC display screen, by using an axis type parameter.
For example, specify P2 for the control axis which is displayed second.
For a spindle type also, specify the order of an axis to be display on the CNC display screen.
A custom macro variable can be used as the value of P.

- Input format of pitch error compensation data (L1)

When the input format of pitch error compensation data is incremental value type (bit 0 (APE) of parameter No. 3602 is set to 0), L1 is not to be specified.
When the input format of pitch error compensation data is total value type (bit 0 (APE) of parameter No. 3602 is set to 1), L1 needs to be specified.
If the data with L1 is input when bit 0 (APE) of parameter No. 3602 is set to 0, or the data without L1 is input when bit 0 (APE) of parameter No. 3602 is set to 1, alarm PS1300 “ILLEGAL ADDRESS” is caused.

⚠ WARNING
1 Do not fail to perform reference position return manually after changing the pitch error compensation data or backlash compensation data. Without this, the machine position can deviate from the correct position.
2 If the compensation data is changed when the input format is total value type(bit 0 (APE) of parameter No. 3602 is set to 1), power must be turned off before operation is continued.
3 The canned cycle mode must be cancelled before entering of parameters. When not cancelled, the drilling motion may be activated.

⚠ CAUTION
Compatibility with the Series 16*i*/18*i*/21*i*:
This model has parameters that are not compatible with the Series 16*i*/18*i*/21*i*. So, before using this function, make a check according to the Parameter Manual (B-64490EN) of this model.

NOTE
Other NC statements cannot be specified while in parameter input mode.

- Writing to a parameter for another path

Programmable parameter input (G10L52) enables writing to a parameter for another path by specifying that path number in either of the following ways:

- Adding a path number to a parameter number (example 4)
- Specifying a path number using system variable #3018 (example 5)

⚠ CAUTION
If changing a parameter for another path by using this function, be sure to take appropriate measures such as combining the function with an inter-path wait, so that the function is executed while the axis for which to change the parameter is stopped.
[Example]
The following shows an example of changing a parameter for path 2 from path 1, using M100 and M101 as wait M codes.

```
Path 1 program
O0100;
      :
      :
M100P12.....(1)
G10L52;
N200xxxxxP_R_   } (A)
G11;            }
M101P12.....(2)
G10L52;
NxxxxxP_R_      } (B)
G11;            }
    :
M30;
```

```
Path 2 program
O0200;
    :
M100P12.... (1)
M101P12.... (2)
      :
      :
M30;

(1): Waits for the start of writing to a
     parameter from 1st path
(2): Waits for the end of writing to a
     parameter from the 1st path
     (Subsequently, operation is performed
     with the parameter changed in (A).)
```

(A):Change to a parameter for another path
→ Must be made after it is checked that the axis on the path for which to change the parameter is stopped.
(B):Change to a parameter for the local path

Example

1. Set bit 2 (SBP) of bit type parameter No. 3404 (when the bit 4 (G1B) of parameter No. 3454 is set to 0)

```
G10L52 ;              Parameter entry mode
N3404 R 00000100 ;    SBP setting
G11 ;                 Cancel parameter entry mode
```

2. Set bit 2 (SBP) of bit type parameter No. 3404 (when the bit 4 (G1B) of parameter No. 3454 is set to 1)

```
G10L52 ;              Parameter entry mode
N3404 Q2 R 1 ;        SBP setting
G11 ;                 Cancel parameter entry mode
```

3. Change the values for the Z-axis (3rd axis) and A-axis (4th axis) in axis type parameter No. 1322 (the coordinates of stored stroke limit 2 in the positive direction for each axis).
   (When the increment systems for the 3rd and 4th axes are IS-B and millimeter machine, respectively)

```
G10L52 ;              Parameter entry mode
N1322P3R4500 ;        Change the value for the Z-axis to 4.500
N1322PP4R12000 ;      Change the value for the A-axis to 12.000
G11 ;                 Cancel parameter entry mode
```

4. Of axis type parameter No. 01322 for the 2nd path, change the values for the 3rd and 4th axes.
   (When the increment systems for the 3rd and 4th axes are IS-B and metric input, respectively)

```
G10L52;               Parameter entry mode
N20001322P3R4500;     Change the value for the 3rd axis in 2nd path to 4.500
N20001322P4R12000;    Change the value for the 4th axis in 2nd path to 12.000
G11;                  Cancel parameter entry mode
```

5. Write to the 4th axis of parameter No. 01322 for the 2nd path

```
#3018=2;              Set the path subject to parameter writing to the 2nd path
G10L52;               Parameter entry mode
N1322P4R12000;        Change the value for the 4th axis in 2nd path to 12.000
G11;                  Cancel parameter entry mode
#3018= 0 ;            Return to the local path
```

6. Change compensation point numbers 10 and 20 of pitch error compensation. (bit 0 (APE) of parameter No. 3602 is set to 0)

```
G10L50 ;              Pitch error compensation data entry mode
N10010R1 ;            Change the compensation point number from 10 to 1
N10020R5 ;            Change the compensation point number from 20 to 5
G11 ;                 Pitch error compensation data entry mode
```

7. Change compensation point numbers 10 and 20 of pitch error compensation. (bit 0 (APE) of parameter No. 3602 is set to 1)

```
G10L50 ;              Pitch error compensation data entry mode
N10010L1R1 ;          Change the compensation point number from 10 to 1
N10020L1R5 ;          Change the compensation point number from 20 to 5
G11 ;                 Pitch error compensation data entry mode
```

19 PATTERN DATA INPUT

Chapter 19, "PATTERN DATA INPUT", consists of the following sections:

19.1 OVERVIEW 542
19.2 EXPLANATION.....542
19.3 EXPLANATION OF OPERATION.....546
19.4 DEFINITION OF THE SCREEN.....550

19.1 OVERVIEW

In the program of the fixed form processing with the custom macro, the operator select the processing pattern on the menu screen and specified the size, number and so on to the variable on the custom macro screen. As above mentioned, this function enables users to perform programming simply without programming using an existing NC language.
With the aid of this function, a machine tool builder can prepare the program of a hole machining cycle (such as a boring cycle or tapping cycle) using the custom macro function, and can store it into the program memory.
This cycle is assigned pattern names, such as BOR1, TAP3, and DRL2.
An operator can select a pattern from the menu of pattern names displayed on the screen.
Data (pattern data) which is to be specified by the operator should be created in advance with variables in a drilling cycle.
The operator can identify these variables using names such as DEPTH, RETURN RELIEF, FEED, MATERIAL or other pattern data names. The operator assigns values (pattern data) to these names.
The operator selects the pattern on the menu screen, and the selected pattern number is assigned to the system variable. The custom macro of the selected pattern can be started by starting a program then referring to the system variable in the program.

19.2 EXPLANATION

This function is consist of Pattern menu screen and Custom macro screen.
The process pattern is selected on the pattern menu screen.
Then the process pattern is selected, the custom macro screen is displayed.
On this custom macro screen, the variable with the name and comment is displayed according to the selected process pattern.
The process data can be input by referring to the variable name with the numerical value on the drawing.

The Fig. 19.2 (a) to Fig. 19.2 (e) are the example for the pattern menu and the custom macro.

(1) Pattern menu screen

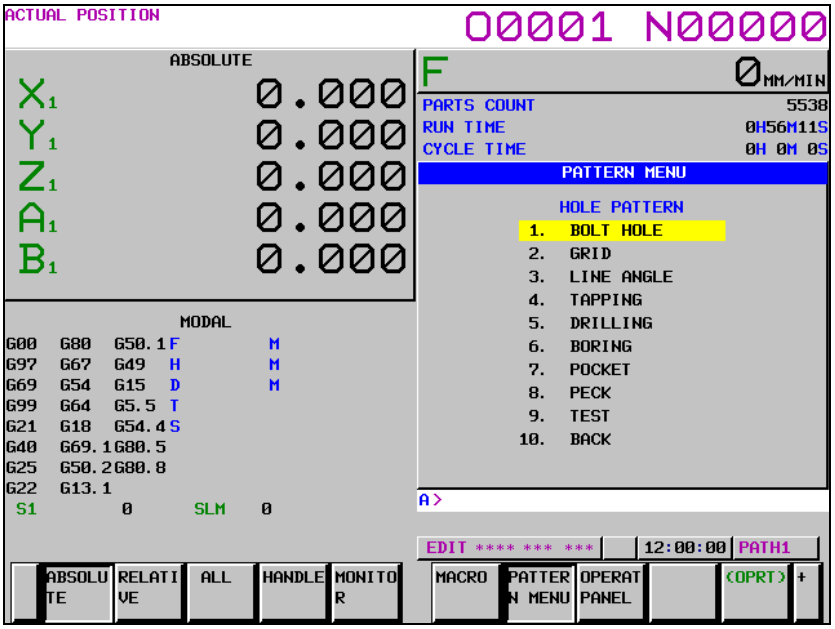


Fig. 19.2 (a) Pattern data menu screen (10.4-inch display unit)

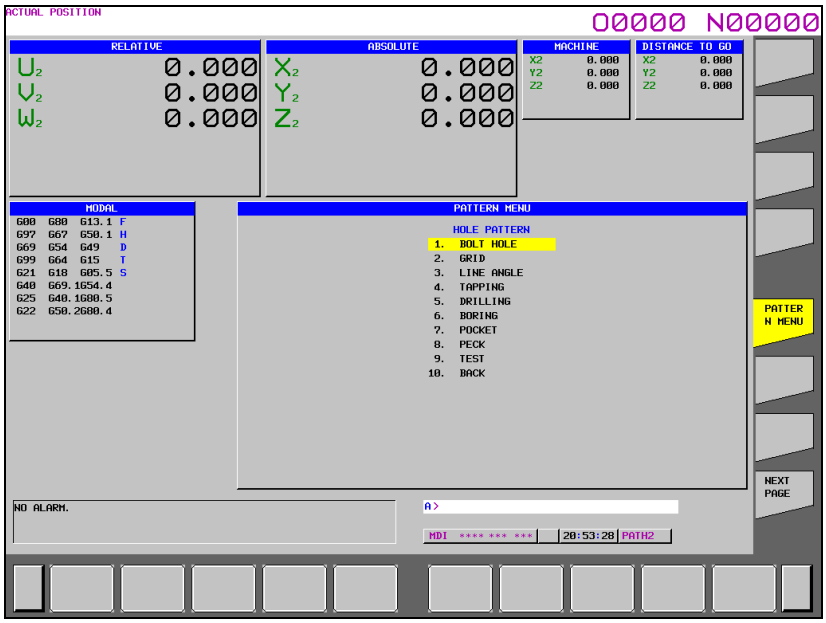


Fig. 19.2 (b) Pattern data menu screen (15-inch display unit)

(2) Custom macro screen

The name of variable and comment can be displayed on the usual custom macro screen.
The menu title and pattern name on the pattern menu screen and the variable name on the custom macro screen can be defined
The position of the comment displayed on the custom macro screen can be selected by setting bit 0 (POC) of parameter No. 11318.
On 15-/19-inch display unit, the comment is displayed at only bottom of the custom macro screen.

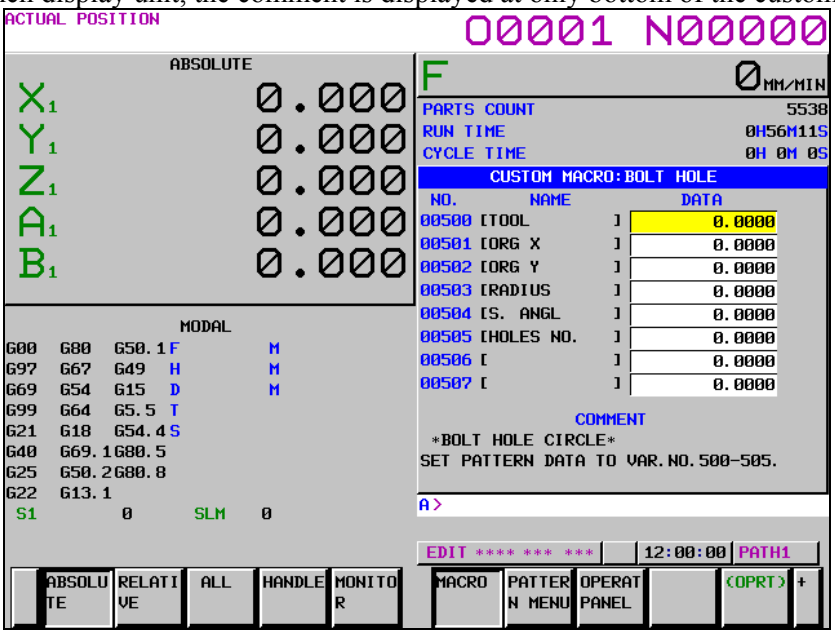


Fig. 19.2 (c) Custom macro screen（bit 0 (POC) of parameter No. 11318=0）(10.4-inch display unit)

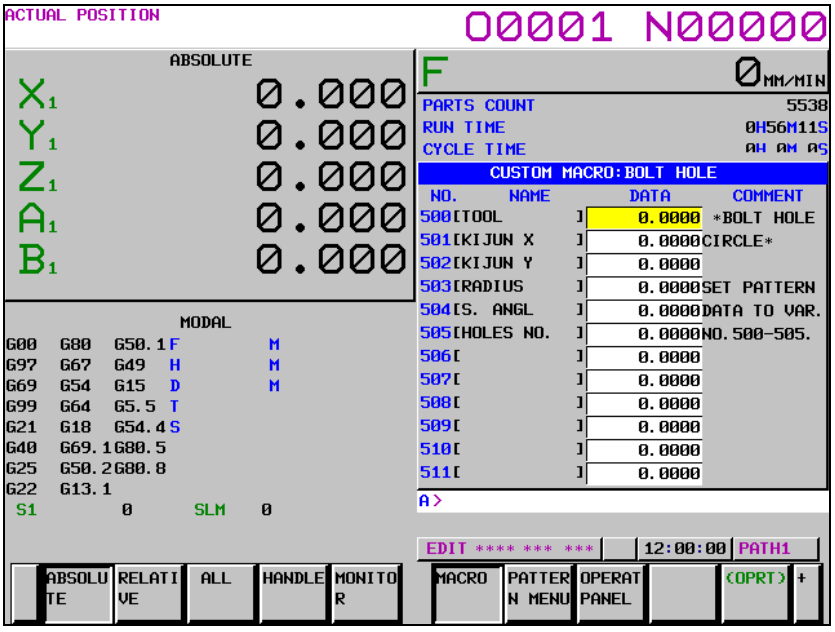


Fig. 19.2 (d) Custom macro screen (bit 0 (POC) of parameter No. 11318=1) (10.4-inch display unit)

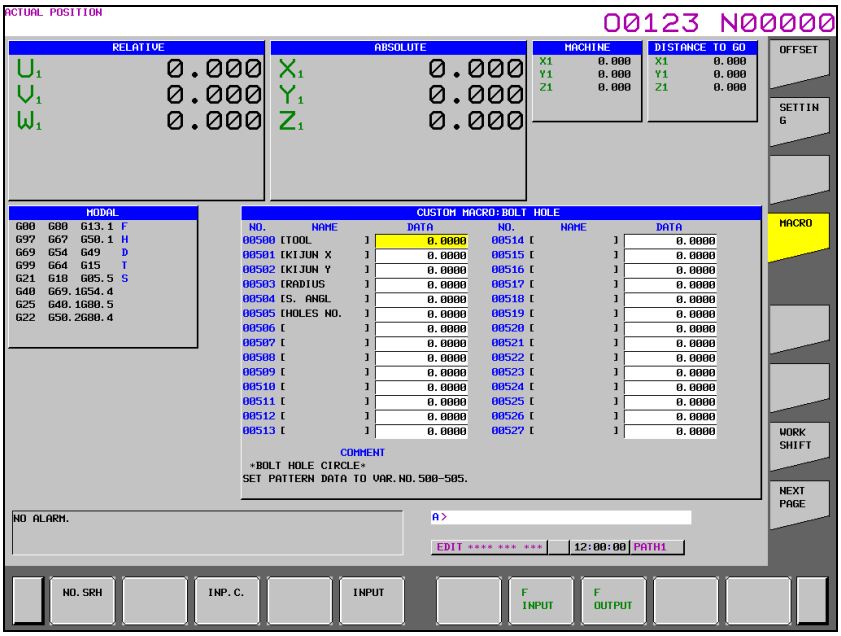


Fig. 19.2 (e) Custom macro screen (15-inch display unit)

19.3 EXPLANATION OF OPERATION

The following explains how to display the pattern menu screen.

For 8.4-/10.4-inch display unit

1 Press function key .
2 Press continuous menu key .
3 Press soft key [PATTERN MENU].

For a 15-/19-inch display unit

1 Press function key .
2 Press vertical soft key [NEXT PAGE] several times.
3 Press vertical soft key [PATTERN MENU].

Pattern menu screen

The pattern menu screen as Fig. 19.3 (a) or Fig. 19.3 (b) is displayed.

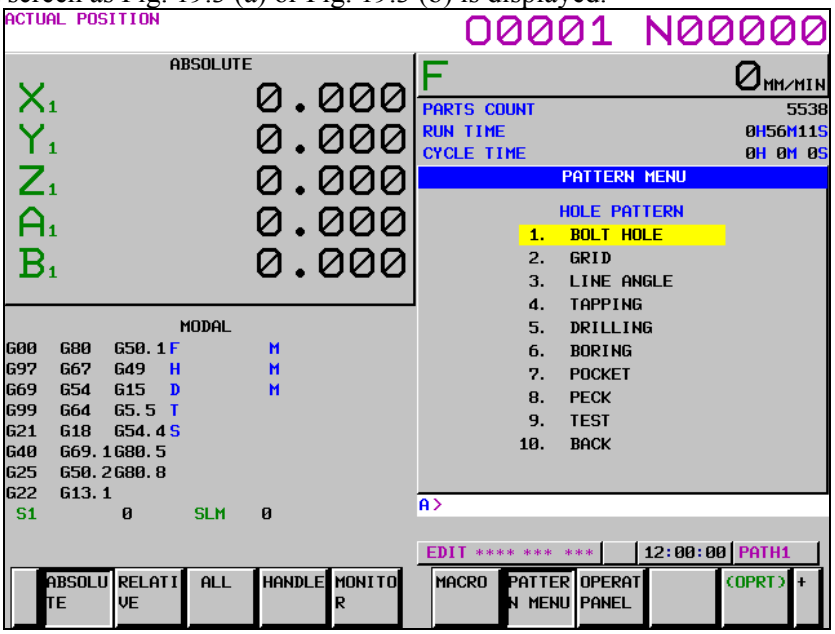


Fig. 19.3 (a) Pattern menu screen (10.4-inch display unit)

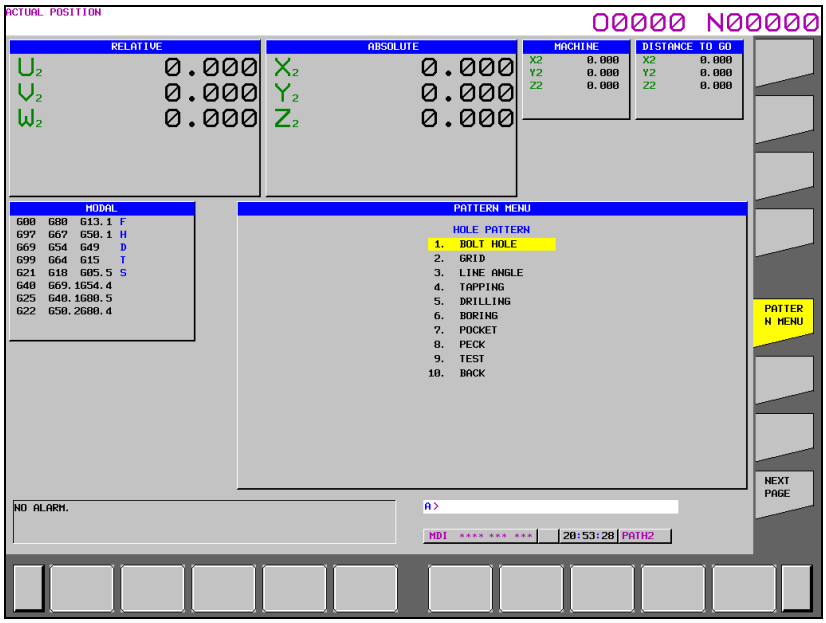


Fig. 19.3 (b) Pattern menu screen (15-inch display unit)

Select the pattern on this screen
The following two methods are effective.

- Selection by cursor

  Move the cursor to the pattern name with the cursor move keys [↑], [↓] and press the soft key [SELECT] or [INPUT].

- Selection by setting of pattern number

  The number that is displayed in the left side of the pattern name is input and press the soft key [select] or [INPUT].

The selected pattern number is registered to system variable #5900. The custom macro of the selected pattern can be started by starting a fixed program (external program No. search) with an external signal. This program refers to the system variable #5900 in the program. This system variable #5900 is kept after power-off.

Custom macro variable screen

The custom macro screen as Fig. 19.3 (c) or Fig. 19.3 (d) is displayed.

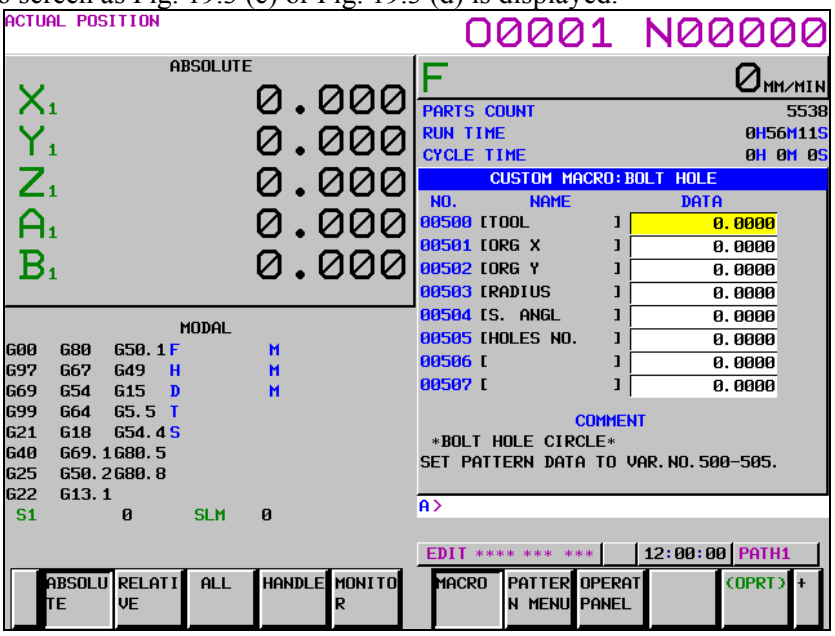


Fig. 19.3 (c) Custom macro screen when the pattern data is input (10.4-inch display unit)

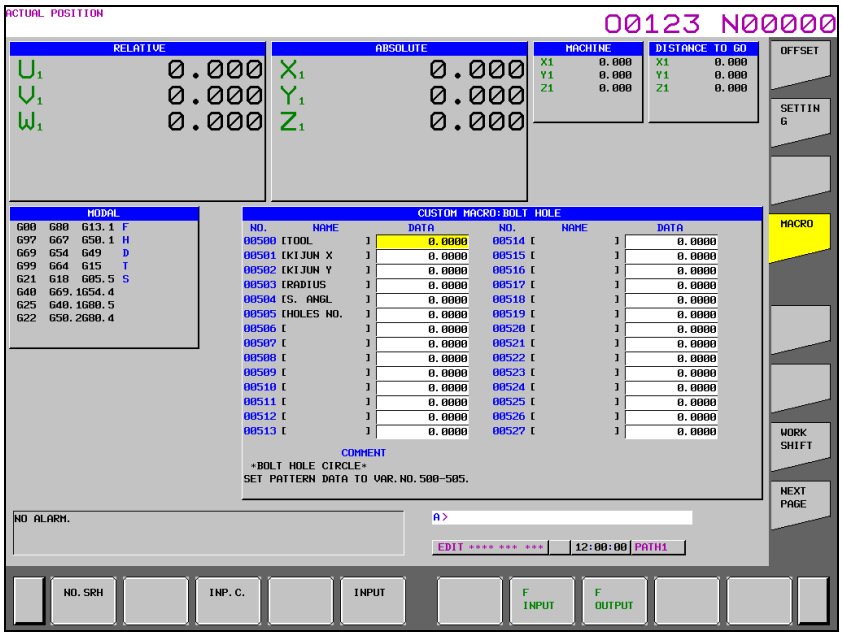


Fig. 19.3 (d) Custom macro screen when the pattern data is input (15-inch display unit)

When the screen is changed to the custom macro screen, the macro variable number that is selected first is specified with the parameters Nos.6101 to 6110. The macro variables that variable name is not defined can be input, too.

NOTE

1 The variable name that is displayed cannot be used as the common variable name of the NC program.
2 When the common variable name is defined by SETVN command, the variable name defined by pattern data input function is given priority.

NOTE

3 When bit 0 (POC) of parameter No.11318 is set to “1”, The variable number is three digit display.
And the value of 12 digits or more is input, 11 digits from head of value are displayed.
Example)
Input: -123456789.123 → Display: -123456789.1

19.4 DEFINITION OF THE SCREEN

The definition of the screen is performed by NC program.

Program configuration

This function is consist of one program for the definition of pattern menu screen and maximum ten programs for the definition of custom macro screen.
Register the macro program for the pattern data input function to the following folder.
//CNC_MEM/USER/LIBRARY/PATH1 to PATH[n]

* [n] is the maximum path number of the system
* The PATH1 to PATH[n] should be made by the machine tool builder.

The program number is as Table 19.4 (a).

Table 19.4 (a) Numbers of subprograms employed in the pattern data input function

Sub program No.	Screen
O9500	Specifies character strings displayed on the pattern data menu.
O9501	Specifies a character string of the pattern data corresponding to pattern No.1
O9502	Specifies a character string of the pattern data corresponding to pattern No.2
O9503	Specifies a character string of the pattern data corresponding to pattern No.3
O9504	Specifies a character string of the pattern data corresponding to pattern No.4
O9505	Specifies a character string of the pattern data corresponding to pattern No.5
O9506	Specifies a character string of the pattern data corresponding to pattern No.6
O9507	Specifies a character string of the pattern data corresponding to pattern No.7
O9508	Specifies a character string of the pattern data corresponding to pattern No.8
O9509	Specifies a character string of the pattern data corresponding to pattern No.9
O9510	Specifies a character string of the pattern data corresponding to pattern No.10

Table 19.4 (b) Macro commands used in the pattern data input function

G code	H code	Function
G65	H90	Specifies the menu title.
G65	H91	Specifies the pattern name.
G65	H92	Specifies the pattern data title.
G65	H93	Specifies the variable name.
G65	H94	Specifies the comment.

Table 19.4 (c) System variables employed in the pattern data input function

System variable	Function
#5900	Pattern No. selected by user.

19.4.1 Definition of the Pattern Menu Screen

Menu title and pattern name are defined as follows.

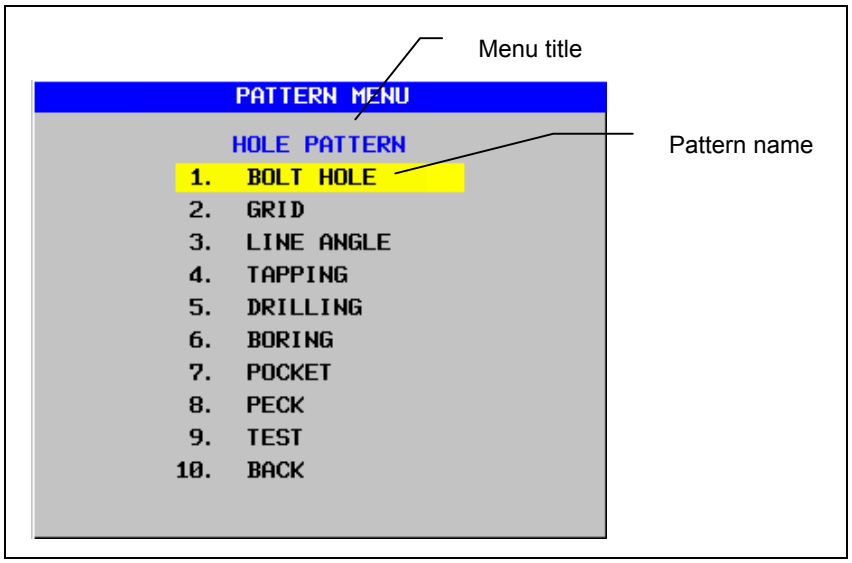


Fig. 19.4.1 (a) Pattern menu screen

Definition of menu title

The character string displayed in the menu title of the pattern menu screen is defined.
The menu title is specified up to 12 characters in a half size letter and up to 6 characters in a full size letter such as kanji character.

- Format

G65 H90 P_ Q_ R_ I_ J_ K_ ;

H90 : Specifies the menu title
P_ : The code of 1st and 2nd characters of title
Q_ : The code of 3rd and 4th characters of title
R_ : The code of 5th and 6th characters of title
I_ : The code of 7th and 8th characters of title
J_ : The code of 9th and 10th characters of title
K_ : The code of 11th and 12th characters of title

As for the way of setting the character-code, refer to the Subsection 19.4.3, "Setting the Character-codes".

Definition of pattern name

The character string displayed in the pattern name which becomes a menu item is defined.
The pattern name is specified up to 10 characters in a half size letter and up to 5 characters in a full size letter.

- **Format**

G65 H91 P_ Q_ R_ I_ J_ K_ ;

H91 : Specifies the pattern name
P_ : Specifies the menu number of the pattern name
The menu number = 1 to 10
Q_ : The code of 1st and 2nd characters of pattern name
R_ : The code of 3rd and 4th characters of pattern name
I_ : The code of 5th and 6th characters of pattern name
J_ : The code of 7th and 8th characters of pattern name
K_ : The code of 9thd and 10th characters of pattern name
As for the way of setting the character-code, refer to the Subsection 19.4.3, "Setting the Character-codes".

Example

The following is example for pattern menu screen.

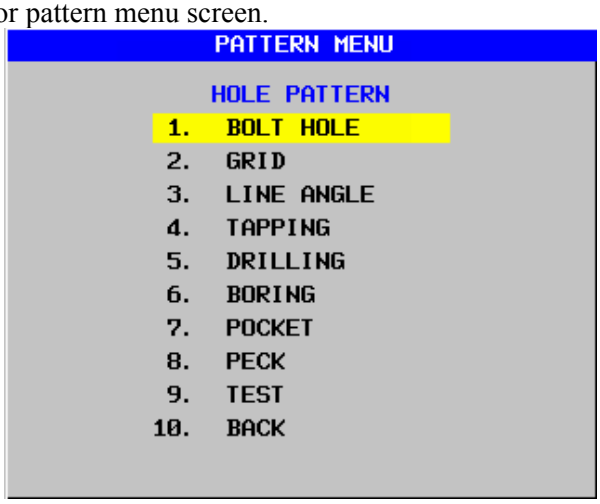


Fig. 19.4.1 (b) Pattern menu screen

```
O9500
N1 G65 H90 P072079 Q076069 R032080 I065084 J084069 K082078;... "HOLE PATTERN"
N2 G65 H91 P1 Q066079 R076084 I032072 J079076 K069032; ..... "BOLT HOLE"
N3 G65 H91 P2 Q071082 R073068; ..... "GRID"
N4 G65 H91 P3 Q076073 R078069 I032065 J078071 K076069; ..... "LINE ANGLE"
N5 G65 H91 P4 Q084065 R080080 I073078 J071032;  ..... "TAPPING"
N6 G65 H91 P5 Q068082 R073076 I076073 J078071;..... "DORILLING"
N7 G65 H91 P6 Q066079 R082073 I078071; ..... "BORING"
N8 G65 H91 P7 Q080079 R067075 I069084; ..... "POCKET"
N9 G65 H91 P8 Q080069 R067075; ..... "PECK"
N10 G65 H91 P9 Q084069 R083084; ..... "TEST"
N11 G65 H91 P10 Q066065 R067075; ..... "BACK"
N12 M99;
```

19.4.2 Definition of the Custom Macro Screen

The title, variable name and comment are defined as follows.

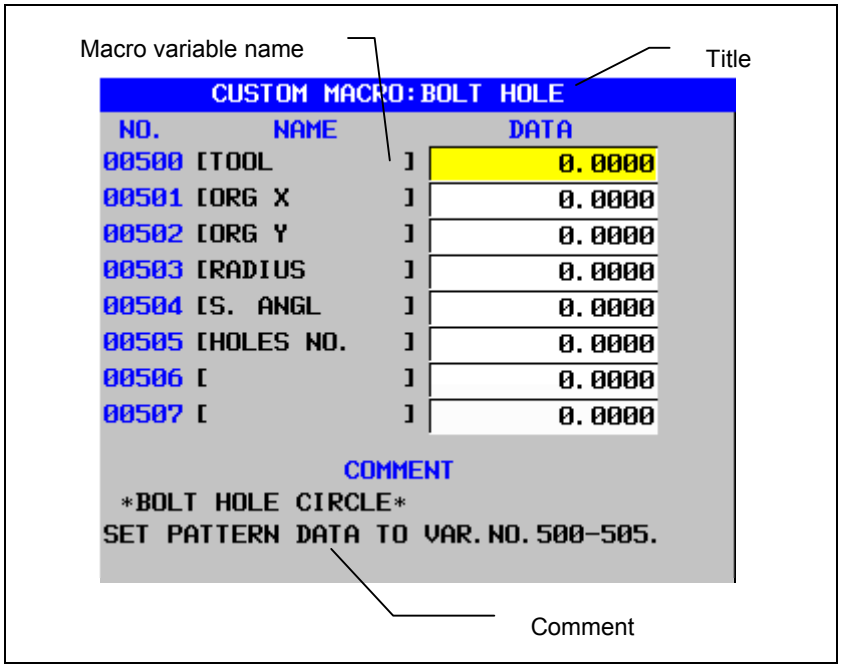


Fig. 19.4.2 (a) Custom macro screen

Definition of title

The character string displayed in the title of the custom macro screen is defined.
The title is specified up to 12 characters in a half size letter and up to 6 characters in a full size letter.

- Format

G65 H92 P_ Q_ R_ I_ J_ K_ ;

H92: Specifies the menu title
P_ : The code of 1st and 2nd characters of the menu title
Q_ : The code of 3rd and 4th characters of the menu title
R_ : The code of 5th and 6th characters of the menu title
I_ : The code of 7th and 8th characters of the menu title
J_ : The code of 9th and 10th characters of the menu title
K_ : The code of 11th and 12th characters of the menu title
As for the way of setting the character-code, refer to the Subsection 19.4.3, "Setting the Character-codes".

Definition of macro variable

The character string displayed in the macro variable name is defined.
The macro variable is specified up to 10 characters in a half size letter and up to 5 characters in a full size letter.

The variable which can be used is as follows
#100 to 199 (100 variables)
#500 to 999 (500 variables), 600 variables in total

- Format

G65 H93 P_ Q_ R_ I_ J_ K_ ;

H93: Specifies the variable name
P_ : Specifies the variable number
Specifies 100 to 199 or 500 to 999
Q_ : The code of 1st and 2nd characters of the variable name
R_ : The code of 3rd and 4th characters of the variable name
I_ : The code of 5th and 6th characters of the variable name
J_ : The code of 7th and 8th characters of the variable name
K_ : The code of 9th and 10th characters of the variable name
As for the way of setting the character-code, refer to the Subsection 19.4.3, "Setting the Character-codes".

Definition of a comment

The character string of the comment displayed on the custom macro screen is defined.
The comment is specified by up to 12 characters in a half size letter and up to 6 characters in a full size letter per one block.

- When the comment is displayed at the bottom（bit 0 (POC) of parameter No.11318=0）
  The number of blocks is defined up to maximum 9 blocks.
  One line is composition of 3 blocks and maximum 3 lines can be displayed.

  The comment is displayed in the order specified by program as the Fig. 19.4.2 (b).
  The block is arranged in defined order.

Block 1	Block 2	Block 3
Block 4	Block 5	Block 6
Block 7	Block 8	Block 9

Fig. 19.4.2 (b) Block layout of comment

- When the comment is displayed at the right（bit 0 (POC) of parameter No.11318=1）
  1 line is composed by 1 blocks, the maximum number of lines is 8 on the 8.4-inch display unit or 12 on the 10.4-inch display unit.

- Format

G65 H94 P_ Q_ R_ I_ J_ K_ ;

H94: Specifies the comment
P_ : The code of 1st and 2nd characters of comment
Q_ : The code of 3rd and 4th characters of comment
R_ : The code of 5th and 6th characters of comment
I_ : The code of 7th and 8th characters of comment
J_ : The code of 9th and 10th characters of comment
K_ : The code of 11th and 12th characters of comment
As for the way of setting the character-code, refer to the Subsection 19.4.3, "Setting the Character-codes".

Example

The following is example of the custom macro screen.

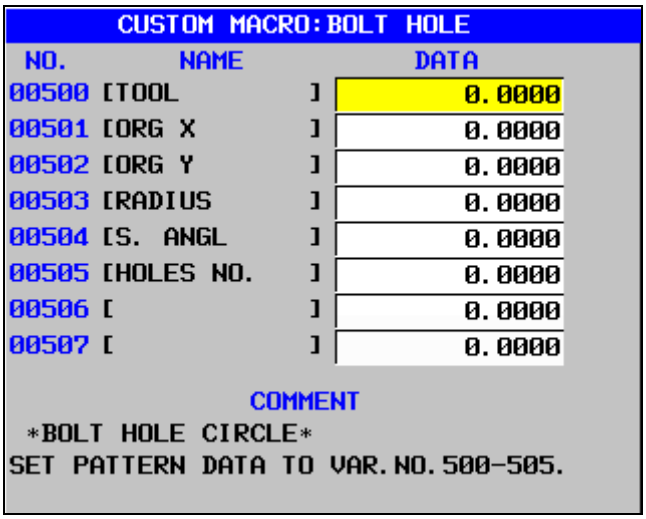


Fig. 19.4.2 (c) Custom macro screen（bit 0 (POC) of parameter No. 11318=0）

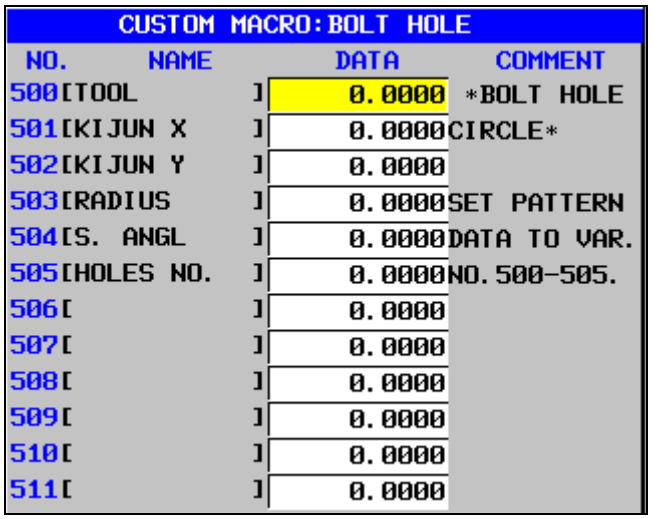


Fig. 19.4.2 (d) Custom macro screen（bit 0 (POC) of parameter No. 11318=1）

```
O9501;
N1 G65 H92 P066079 Q076084 R032072 I079076 J069032;.....("BOLT HOLE")
N2 G65 H93 P500 Q084079 R079076; .....("TOOL")
N3 G65 H93 P501 Q079082 R071032 I08832; .....("ORG X")
N4 G65 H93 P502 Q079082 R071032 I08932; .....("ORG Y")
N5 G65 H93 P503 Q082065 R068073 I085803; .....("RADIUS")
N6 G65 H93 P504 Q083046 R032065 I078071 J076032.....("S. ANGL")
N7 G65 H93 P505 Q072079 R076079 I083032 J078079 K046032.....("HOLES NO.")
N8 G65 H94 P032042 Q066079 R076084 I032072 J079076 K069032;.....(" *BOLT HOLE")
N9 G65 H94 P067073 Q082067 R076069 I042032; .....("CIRCLE*")
N10 G65 H94 P083069 Q084032 R080065 I084084 J069082 K078032;...("SET PATTERN")
N11 G65 H94 P068065 Q084065 R032084 I079032 J086065 K082046;...("DATA NO VAR.")
N12 G65 H94 P078079 Q046053 R048048 I045053 J048053 K046032;...("NO500-505")
N13 M99;
```

19.4.3 Setting the Character-codes

The character cannot be used to specify the NC program.
Therefore, the code corresponding to the character is specified.
One character is consist of three figures in a half size letter and six figures in a full size letter.
The character code is specified for each address of the G65 instruction by six digits.

Refer to the table for the character code.
Example)
When "ABCDEFGH" is specified, the description of the code is as follows.
Encoded character string : 065 066 067 068 069 070 071 072

```
P065066 Q067068 R069070 I071072;
  AB      CD      EF      GH
```

NOTE
1 Space (032) is added ahead of the character-code, when the character-code of three digits or less is specified.
Example)
P065066 Q067; → " AB C "
032(space) is put at the end, when "ABC" is displayed.
P065066 Q067032; → " ABC "
2 It is assumed in that the space of two characters was defined in the address when there is an address not defined.
Example)
P065066 I067068; → "AB CD"

Characters and codes to be used for the pattern data input function

Character	Code	Comment	Character	Code	Comment
A	065		6	054	
B	066		7	055	
C	067		8	056	
D	068		9	057	
E	069			032	Space
F	070		!	033	Exclamation mark
G	071		“	034	Quotation mark
H	072		#	035	Hash sign
I	073		$	036	Dollar sign
J	074		%	037	Percent
K	075		&	038	Ampersand
L	076		‘	039	Apostrophe
M	077		*	042	Asterisk
N	078		+	043	Plus sign
O	079		,	044	Comma
P	080		-	045	Minus sign
Q	081		.	046	Period
R	082		/	047	Slash
S	083		:	058	Colon
T	084		;	059	Semicolon
U	085		<	060	Left angle bracket
V	086		=	061	Equal sign
W	087		>	062	Right angle bracket
X	088		?	063	Question mark
Y	089		@	064	At mark

Character	Code	Comment	Character	Code	Comment
Z	090		[	091	Left square bracket
0	048		¥	092	Yen sign
1	049		]	093	Right square bracket
2	050		^	094	
3	051		_	095	Underscore
4	052				
5	053				

The characters and the codes of the katakana is as follows.

Character	Code	Comment	Character	Code	Comment
ア	177		ム	209	
イ	178		メ	210	
ウ	179		モ	211	
エ	180		ヤ	212	
オ	181		ユ	213	
カ	182		ヨ	214	
キ	183		ラ	215	
ク	184		リ	216	
ケ	185		ル	217	
コ	186		レ	218	
サ	187		ロ	219	
シ	188		ワ	220	
ス	189		ヲ	166	
セ	190		ン	221	
ソ	191		ァ	167	
タ	192		ィ	168	
チ	193		ゥ	169	
ツ	194		ェ	170	
テ	195		ォ	171	
ト	196		ャ	172	
ナ	197		ュ	173	
ニ	198		ョ	174	
ヌ	199		ッ	175	
ネ	200		″	222	Diacritical mark
ノ	201		°	223	Diacritical mark
ハ	202		。	161	Punctuation
ヒ	203		「	162	Left quotation mark
フ	204		」	163	Right quotation mark
ヘ	205		、	164	Comma
ホ	206		・	165	Point
マ	207			000	Space
ミ	208				

NOTE
Diacritical mark is one character.

The characters and the codes of the hiragana and the kanji are as follows. The following hiraganas and kanjis use two characters of the alphanumeric character.

ぁ	あ	ぃ	い	う	う	ぇ	え	ぉ	お
002 000	002 002	002 004	002 006	002 008	002 010	002 012	002 014	002 016	002 018
か	が	き	ぎ	く	ぐ	け	げ	こ	ご
002 020	002 022	002 024	002 026	002 028	002 030	002 032	002 034	002 036	002 038

さ	ざ	し	じ	す	ず	せ	ぜ	そ	ぞ
002 040	002 042	002 044	002 046	002 048	002 050	002 052	002 054	002 056	002 058
た	だ	ち	ぢ	っ	つ	づ	て	で	と
002 060	002 062	002 064	002 066	002 068	002 070	002 072	002 074	002 076	002 078
ど	な	に	ぬ	ね	の	は	ば	ぱ	ひ
002 080	002 082	002 084	002 086	002 088	002 090	002 092	002 094	002 096	002 098
び	ぴ	ふ	ぶ	ぷ	へ	べ	ぺ	ほ	ぼ
002 100	002 102	002 104	002 106	002 108	002 110	002 112	002 114	002 116	002 118
ぽ	ま	み	む	め	も	ゃ	や	ゅ	ゆ
002 120	002 122	002 124	002 126	002 128	002 130	002 132	002 134	002 136	002 138
ょ	よ	ら	り	る	れ	ろ	わ	わ	素
002 140	002 142	002 144	002 146	002 148	002 150	002 152	002 154	002 156	002 158
材	を	ん	種	類	棒	穴	成	形	質
002 160	002 162	002 164	002 166	002 168	002 170	002 172	002 174	002 176	002 178
寸	法	外	径	長	端	面	最	小	内
002 180	002 182	002 184	002 186	002 188	002 190	002 192	002 194	002 196	002 198
大	加	工	切	削	倣	正	途	中	荒
002 200	002 202	002 204	002 206	002 208	002 210	002 212	002 214	002 216	002 218
具	番	号	仕	上	込	点	方	向	速
002 220	002 222	002 224	002 226	002 228	002 230	002 232	002 234	002 236	002 238
度	送	量	開	始	深	主	軸		
002 240	002 242	002 244	002 246	002 248	002 250	002 252	002 254		
回	転	数	位	置	決	直	線	時	円
003 000	003 002	003 004	003 006	003 008	003 010	003 012	003 014	003 016	003 018
反	現	在	指	令	値	領	域	診	断
003 020	003 022	003 024	003 026	003 028	003 030	003 032	003 034	003 036	003 038
操	作	手	引	機	械	残	移	動	次
003 040	003 042	003 044	003 046	003 048	003 050	003 052	003 054	003 056	003 058
早	電	源	投	入	間	分	秒	自	運
003 060	003 062	003 064	003 066	003 068	003 070	003 072	003 074	003 076	003 078
負	荷	実	使	用	寿	命	新	規	除
003 080	003 082	003 084	003 086	003 088	003 090	003 092	003 094	003 096	003 098
隅	取	単	補	能	独	終	了	記	角
003 100	003 102	003 104	003 106	003 108	003 110	003 112	003 114	003 116	003 118
溝	刃	幅	広	設	定	ー	覧	表	部
003 120	003 122	003 124	003 126	003 128	003 130	003 132	003 134	003 136	003 138
炭	合	金	鋼	超	硬	先	付	摩	耗
003 140	003 142	003 144	003 146	003 148	003 150	003 152	003 154	003 156	003 158
仮	想	副	行	挿	消	去	山	高	準
003 160	003 162	003 164	003 166	003 168	003 170	003 172	003 174	003 176	003 178
備	完	後	弧	助	択	無	視	器	原
003 180	003 182	003 184	003 186	003 188	003 190	003 192	003 194	003 196	003 198
登	録	再	処	理	描	画	過	容	編
003 200	003 202	003 204	003 206	003 208	003 210	003 212	003 214	003 216	003 218
集	未	対	相	座	標	示	名	歯	変
003 220	003 222	003 224	003 226	003 228	003 230	003 232	003 234	003 236	003 238
呼	推	馬	力	系	選	達	閉		
003 240	003 242	003 244	003 246	003 248	003 250	003 252	003 254		
禁	復	帰	書	個	桁	稼	由	両	半
004 000	004 002	004 004	004 006	004 008	004 010	004 012	004 014	004 016	004 018
逃	底	逆	下	空	四	触	平	代	辺
004 020	004 022	004 024	004 026	004 028	004 030	004 032	004 034	004 036	004 038

格	子	周	心	本	群	停	止	巾	微
004 040	004 042	004 044	004 046	004 048	004 050	004 052	004 054	004 056	004 058
状	路	範	囲	倍	率	注	側	特	殊
004 060	004 062	004 064	004 066	004 068	004 070	004 072	004 074	004 076	004 078
距	離	連	続	増	隔	件	初	期	条
004 080	004 082	004 084	004 086	004 088	004 090	004 092	004 094	004 096	004 098
経	握	圧	扱	陰	隠	右	押	横	黄
004 100	004 102	004 104	004 106	004 108	004 110	004 112	004 114	004 116	004 118
億	屋	化	何	絵	階	概	該	巻	換
004 120	004 122	004 124	004 126	004 128	004 130	004 132	004 134	004 136	004 138
気	起	軌	技	疑	供	共	境	強	教
004 140	004 142	004 144	004 146	004 148	004 150	004 152	004 154	004 156	004 158
掘	繰	係	傾	型	検	権	研	肩	見
004 160	004 162	004 164	004 166	004 168	004 170	004 172	004 174	004 176	004 178
験	元	弦	減	孔	巧	控	更	校	構
004 180	004 182	004 184	004 186	004 188	004 190	004 192	004 194	004 196	004 198
根	左	差	雑	参	散	産	算	治	耳
004 200	004 202	004 204	004 206	004 208	004 210	004 212	004 214	004 216	004 218
式	失	修	十	従	勝	商	少	尚	昇
004 220	004 222	004 224	004 226	004 228	004 230	004 232	004 234	004 236	004 238
植	色	食	伸	信	侵	振	浸		
004 240	004 242	004 244	004 246	004 248	004 250	004 252	004 254		
真	暗	以	意	異	影	鋭	越	価	可
005 000	005 002	005 004	005 006	005 008	005 010	005 012	005 014	005 016	005 018
科	果	箇	課	各	拡	核	学	掛	漢
005 020	005 022	005 024	005 026	005 028	005 030	005 032	005 034	005 036	005 038
簡	観	関	含	却	客	休	急	業	曲
005 040	005 042	005 044	005 046	005 048	005 050	005 052	005 054	005 056	005 058
均	筋	継	計	軽	言	限	互	降	採
005 060	005 062	005 064	005 066	005 068	005 070	005 072	005 074	005 076	005 078
済	細	姿	思	写	射	斜	者	車	借
005 080	005 082	005 084	005 086	005 088	005 090	005 092	005 094	005 096	005 098
縦	重	出	述	術	渉	照	省	章	証
005 100	005 102	005 104	005 106	005 108	005 110	005 112	005 114	005 116	005 118
象	身	進	人	図	違	印	沿	遠	央
005 120	005 122	005 124	005 126	005 128	005 130	005 132	005 134	005 136	005 138
奥	往	応	会	解	改	割	活	願	基
005 140	005 142	005 144	005 146	005 148	005 150	005 152	005 154	005 156	005 158
奇	寄	岐	既	近	区	矩	駆	偶	旧
005 160	005 162	005 164	005 166	005 168	005 170	005 172	005 174	005 176	005 178
求	球	究	級	欠	結	口	語	誤	交
005 180	005 182	005 184	005 186	005 188	005 190	005 192	005 194	005 196	005 198
厚	項	刻	告	黒	財	策	糸	試	資
005 200	005 202	005 204	005 206	005 208	005 210	005 212	005 214	005 216	005 218
事	持	似	釈	弱	受	収	純	順	所
005 220	005 222	005 224	005 226	005 228	005 230	005 232	005 234	005 236	005 238
序	剰	場	常	飾	水	錐	据		
005 240	005 242	005 244	005 246	005 248	005 250	005 252	005 254		
制	整	製	前	全	然	則	属	即	他
006 000	006 002	006 004	006 006	006 008	006 010	006 012	006 014	006 016	006 018
多	存	谷	探	短	徴	鎮	調	頂	鉄
006 020	006 022	006 024	006 026	006 028	006 030	006 032	006 034	006 036	006 038

添	頭	同	導	道	熱	年	濃	箱	発
006 040	006 042	006 044	006 046	006 048	006 050	006 052	006 054	006 056	006 058
抜	伴	必	百	複	物	文	聞	併	忘
006 060	006 062	006 064	006 066	006 068	006 070	006 072	006 074	006 076	006 078
末	密	有	余	与	裏	立	略	青	席
006 080	006 082	006 084	006 086	006 088	006 090	006 092	006 094	006 096	006 098
石	積	赤	接	折	粗	創	双	捜	太
006 100	006 102	006 104	006 106	006 108	006 110	006 112	006 114	006 116	006 118
打	体	待	態	替	段	知	地	致	遅
006 120	006 122	006 124	006 126	006 128	006 130	006 132	006 134	006 136	006 138
追	通	伝	得	読	凸	凹	突	鈍	敗
006 140	006 142	006 144	006 146	006 148	006 150	006 152	006 154	006 156	006 158
杯	背	配	品	不	布	並	頁	別	片
006 160	006 162	006 164	006 166	006 168	006 170	006 172	006 174	006 176	006 178
返	勉	弁	保	明	滅	木	目	歪	揺
006 180	006 182	006 184	006 186	006 188	006 190	006 192	006 194	006 196	006 198
様	溶	要	抑	良	輪	和	話	枠	節
006 200	006 202	006 204	006 206	006 208	006 210	006 212	006 214	006 216	006 218
説	絶	千	専	浅	旋	総	走	退	台
006 220	006 222	006 224	006 226	006 228	006 230	006 232	006 234	006 236	006 238
第	題	卓	室	着	柱	鋳	丁		
006 240	006 242	006 244	006 246	006 248	006 250	006 252	006 254		
低	訂	肉	日	白	薄	比	皮	被	非
007 000	007 002	007 004	007 006	007 008	007 010	007 012	007 014	007 016	007 018
美	普	伏	歩	包	門	問	絡	列	万
007 020	007 022	007 024	007 026	007 028	007 030	007 032	007 034	007 036	007 038
利	訳	礼	乱	放	枚	約	練	油	劣
007 040	007 042	007 044	007 046	007 048	007 050	007 052	007 054	007 056	007 058
例	郭	戻	冷	垂	緑	紫	許	測	精
007 060	007 062	007 064	007 066	007 068	007 070	007 072	007 074	007 076	007 078
効	→	↗	↑	↖	←	↙	↓	↘	
007 080	007 082	007 084	007 086	007 088	007 090	007 092	007 094	007 096	007 098
				板	予	〃	家	装	管
007 100	007 102	007 104	007 106	007 108	007 110	007 112	007 114	007 116	007 118
粉	等					貫	安	α	β
007 120	007 122	007 124	007 126	007 128	007 130	007 132	007 134	007 136	007 138
程	抗	張	任	破	損	御	足	守	般
007 140	007 142	007 144	007 146	007 148	007 150	007 152	007 154	007 156	007 158
納	義	丸	汎	固	毎	当	的	詳	鳥
007 160	007 162	007 164	007 166	007 168	007 170	007 172	007 174	007 176	007 178
適	論	額	縁	温	給	界	混	監	締
007 180	007 182	007 184	007 186	007 188	007 190	007 192	007 194	007 196	007 198
護	己	称	樹	脂	料	落	確	認	報
007 200	007 202	007 204	007 206	007 208	007 210	007 212	007 214	007 216	007 218
排	性	生	績	判	搬	砥	θ	島	壁
007 220	007 222	007 224	007 226	007 228	007 230	007 232	007 234	007 236	007 238
				]	[	▗	■		
007 240	007 242	007 244	007 246	007 248	007 250	007 252	007 254		

20 HIGH-SPEED CUTTING FUNCTIONS

Chapter 20, "HIGH-SPEED CUTTING FUNCTIONS", consists of the following sections:

20.1 AI CONTOUR CONTROL FUNCTION I AND AI CONTOUR CONTROL FUNCTION II (G05.1) 561
20.2 MACHINING CONDITION SELECTING FUNCTION 575
20.3 MACHINING QUALITY LEVEL ADJUSTMENT 575
20.4 HIGH-SPEED CYCLE MACHINING 577
20.5 HIGH-SPEED BINARY PROGRAM OPERATION 578
20.6 OPTIMUM ACCELERATION/DECELERATION FOR RIGID TAPPING 579
20.7 PATH TABLE OPERATION 581

20.1 AI CONTOUR CONTROL FUNCTION I AND AI CONTOUR CONTROL FUNCTION II (G05.1)

Overview

The AI contour control I and AI contour control II functions are provided for high-speed, high-precision machining. This function enables suppression of acceleration/deceleration delays and servo delays that become larger with increases in the feedrate and reduction of machining profile errors.
When a high-speed processing option is used with AI contour control II, high-speed operation processing is enabled, therefore, faster and more precise machining is enabled. In addition, an option to expand read-ahead blocks to up to 1000 blocks is available.
In the descriptions below that are common to AI contour control I and AI contour control II, the term "AI contour control" is used.

Format

```
G05.1 Q1 [R_] ;        AI contour control mode on
  :
G05.1 Q0 ;             AI contour control mode off
  R  :  Machining condition selecting level (1 to 10)
```

The AI contour control mode can be controlled also with the formats that have been used for the conventional advanced preview control, high-precision contour control, and AI high-precision contour control functions.

```
G08 P1 [R_] ;          AI contour control mode on
  :
G08 P0 ;               AI contour control mode off
  R  :  Machining condition selecting level (1 to 10)
```

```
G05 P10000 [R_] ;      AI contour control mode on
  :
G05 P0 ;               AI contour control mode off
  R  :  Machining condition selecting level (1 to 10)
```

NOTE
1 Always specify G05.1, G08, and G05 in an independent block.
（Do not specify other G codes at the same time.）
2 G05 can be specified only for AI contour control II.
3 The AI contour control mode can also be turned off at a reset.
4 The AI contour control mode can be turned on at the start of automatic operation by setting bit 0 (SHP) of parameter No. 1604.

Valid functions

The functions listed below are valid in the AI contour control mode.
Valid functions are limited depending on the command format and whether AI contour control I or II is used.
Refer to the manual supplied by the machine tool builder or contact the machine tool builder for the function etc. that can be actually used.

Function	AI contour control I	AI contour control II	AI contour control II with high-speed processing
Number of blocks read ahead	40 (*1) (When G8 is specified: 1)	200 (When G8 is specified: 1)	600 (*2) (*3) (When G8 is specified: 1)
Speed control with feedrate difference on each axis	Enabled	Enabled	Enabled
Speed control with acceleration in circular interpolation	Enabled	Enabled	Enabled
Speed control with acceleration on each axis	Enabled (When G8 is specified: Not enabled)	Enabled (When G8 is specified: Not enabled)	Enabled (When G8 is specified: Not enabled)
Smooth speed control	Not enabled	Enabled (When G8 is specified: Not enabled)	Enabled (When G8 is specified: Not enabled)
Speed control with cutting load	Not enabled	Enabled	Enabled
Disregard of feedrate command	Not enabled	Enabled	Enabled

*1 In the system of which maximum controllable paths are more than two, the number of blocks is 30.
*2 The number of blocks can optionally be expanded to 1000 blocks.
*3 For a 2-path system, use parameter No. 11604 to set the number of blocks read ahead for each path.

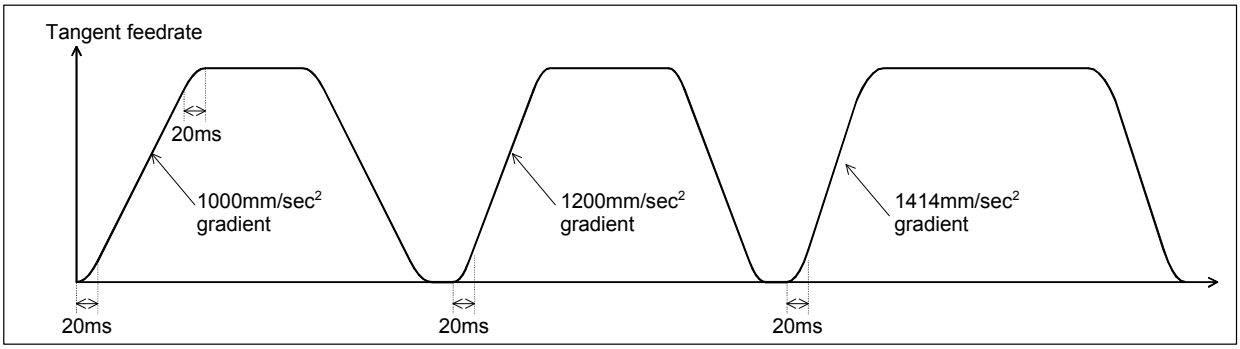
Explanation

- Method of determining the tangent acceleration

Acceleration/deceleration is performed with the largest tangent acceleration/deceleration that does not exceed the acceleration set for each axis.
(Example)
X-axis permissible acceleration: 1000 mm/sec^2
Y-axis permissible acceleration: 1200 mm/sec^2
Acceleration change time: 20 msec
Program:

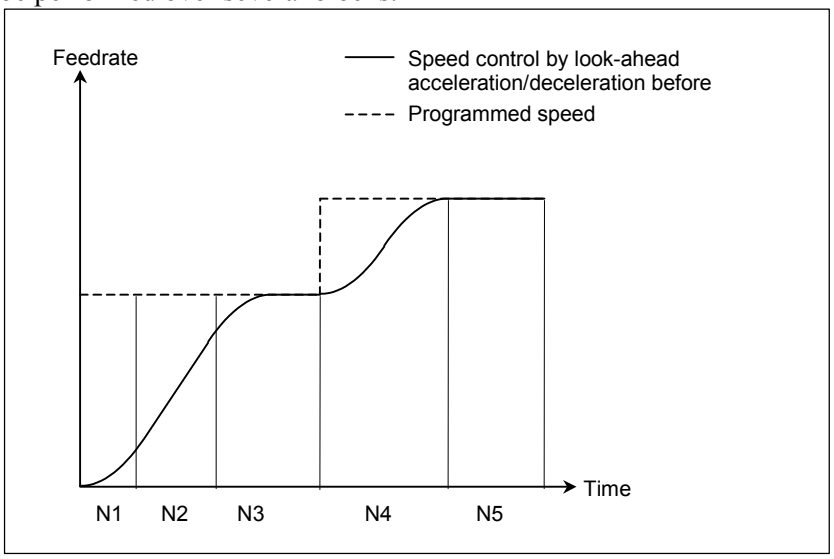
```
N1 G01 G91 X20. F6000     Move on the X-axis.
G04 X0.01
N2 Y20.                   Move on the Y-axis.
G04 X0.01
N3 X20. Y20.              Move in the XY direction (at 45 degrees).
```

The acceleration in N3 is 1414 mm/sec^2. At this point, the acceleration on the X-axis is equal to the set value (1000 mm/sec^2).



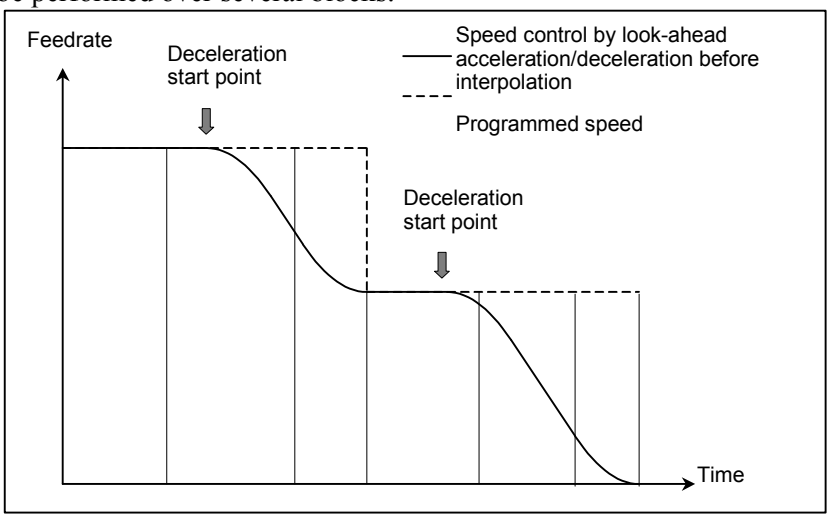
- Acceleration

Acceleration is performed so that the feedrate programmed for a block is attained at the beginning of the block.
Acceleration can be performed over several blocks.



- Deceleration

Deceleration starts in advance so that the feedrate programmed for a block is attained at the beginning of the block.
Deceleration can be performed over several blocks.

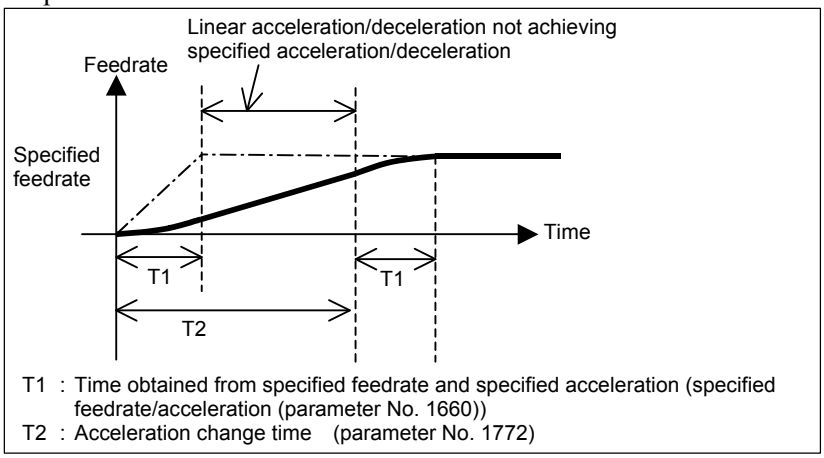


- Deceleration based on a distance

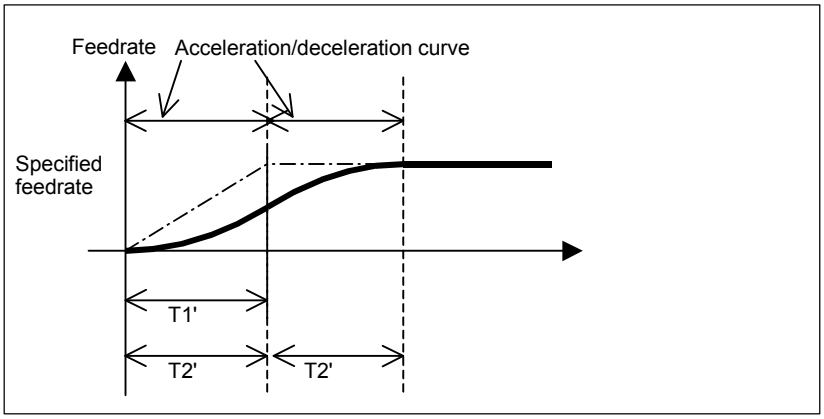
If the total distance of the blocks read ahead becomes shorter than or equal to the deceleration distance obtained from the current feedrate, deceleration starts.
If the total distance of the blocks read ahead during deceleration increases, acceleration is performed.
If the blocks of a small amount of travel are successively specified, deceleration and acceleration may be performed alternately, making the feedrate inconsistent.
To avoid this, decrease the programmed feedrate.

- Function for changing time constant of bell-shaped acceleration/deceleration

Bell-shaped acceleration/deceleration before interpolation is performed using the parameter-set acceleration and acceleration change time. If a low feedrate is specified, a linear acceleration/ deceleration may not achieve the specified acceleration as shown:



In such a case, set bit 3 (BCG) of parameter No. 7055 to 1. Then, the internal acceleration and vector time constant of acceleration/deceleration before interpolation are changed to make the acceleration/deceleration pattern as close as possible to the optimum bell-shaped acceleration/deceleration before interpolation based on a specified acceleration/deceleration reference speed, and so acceleration/deceleration time is reduced.



There are three methods for specifying the acceleration/deceleration reference speed.
(1) Specifying the speed using an F in a G05.1 Q1 block
(2) Setting the speed on parameter No.7066
(3) Setting the speed specified with the F command issued at the start of cutting as the reference speed

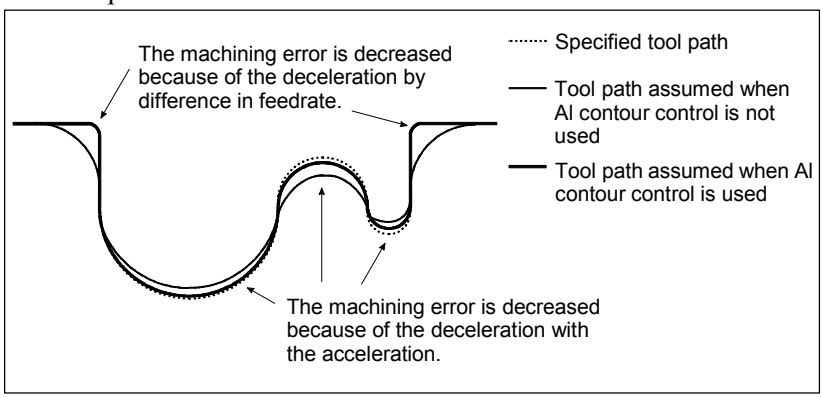
When F is specified in a G05.1Q1 block, the specified feedrate is assumed to be the acceleration/deceleration reference speed. This command can be used only in the feed per minute mode.

If no F command is specified in a G05.1Q1 block, the feedrate specified in parameter No. 7066 is assumed to be the acceleration/deceleration reference speed. If 0 is set in parameter No. 7066, the F command specified in the cutting start block is assumed to be the acceleration/deceleration reference speed.

- Automatic feedrate control function

In AI contour control mode, the feedrate is automatically controlled by the reading-ahead of blocks.
The feedrate is determined using the following conditions. If the specified feedrate exceeds the determined feedrate, acceleration/ deceleration before interpolation is performed to achieve the determined feedrate.

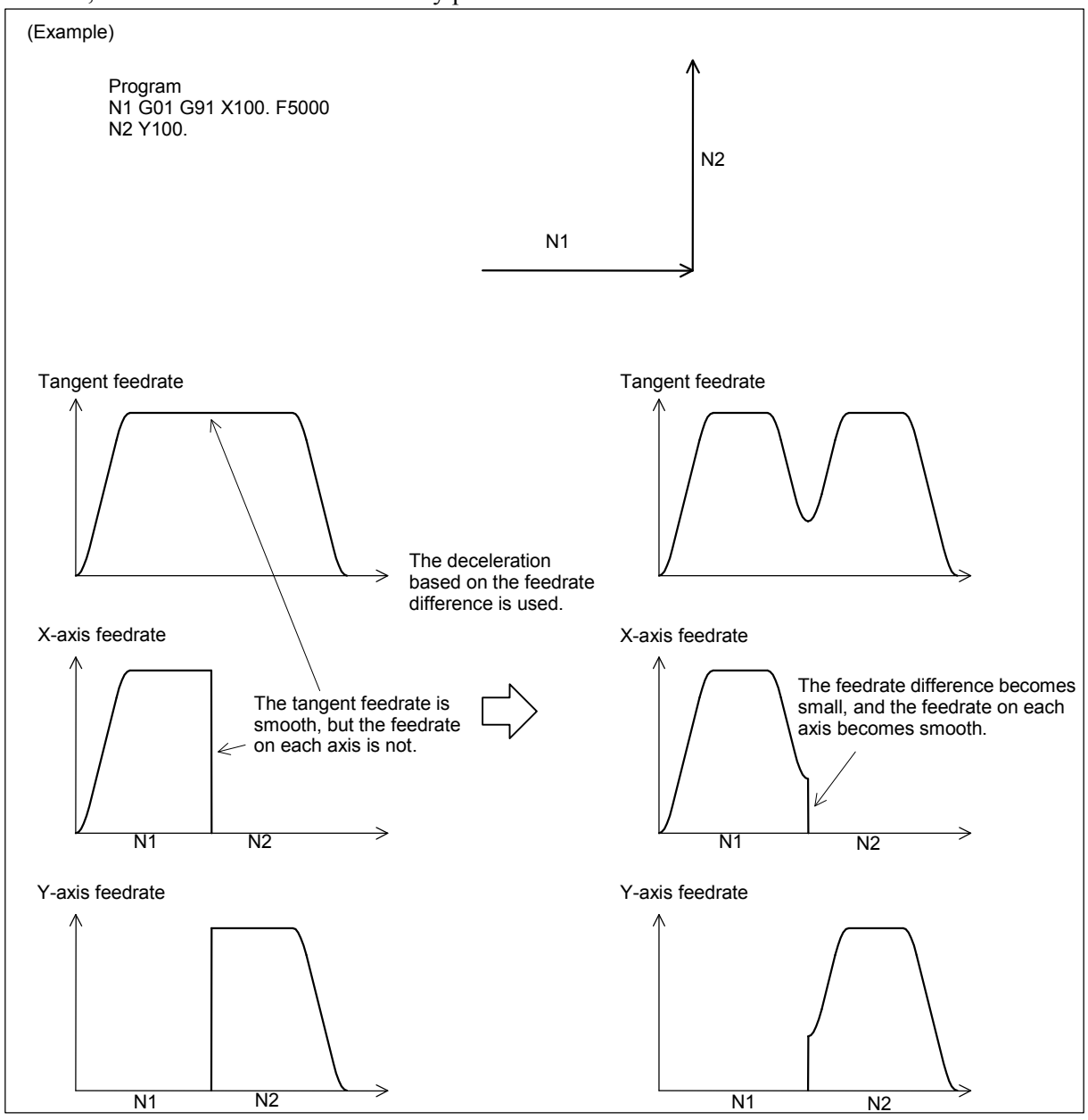
<1> Feedrate changes on each axis at a corner and the permissible feedrate change that has been set
<2> Expected acceleration on each axis and the permissible acceleration that has been set
<3> Cutting load that is expected from the travel direction on the Z-axis



For details, see the explanation of each function.

- Speed control based on the feedrate difference on each axis at a corner

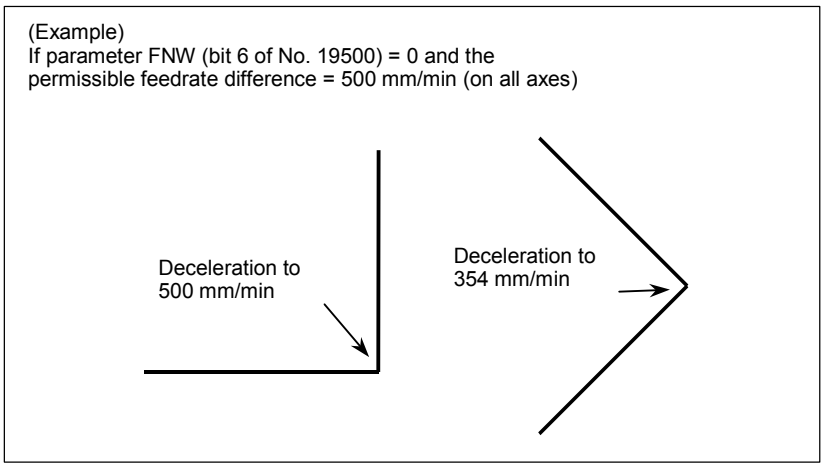
By using the speed control based on the feedrate difference on each axis at a corner, if a feedrate change occurs on an axis on each axis at a corner, the feedrate is determined so that any feedrate difference exceeding the permissible feedrate difference on that axis that has been set for parameter No. 1783 does not occur, and deceleration is automatically performed.



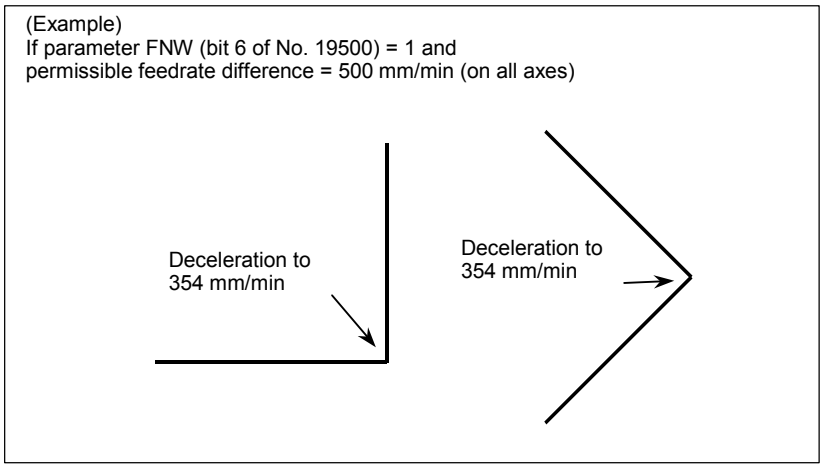
The method of deceleration based on the feedrate difference differs depending on the setting made for bit 6 (FNW) of parameter No. 19500.

If "0" is set, the largest feedrate that does not exceed the permissible feedrate difference set for parameter No. 1783 is assumed to be the deceleration feedrate.

In this case, the deceleration feedrate differs if the travel direction differs, even if the shape is the same.

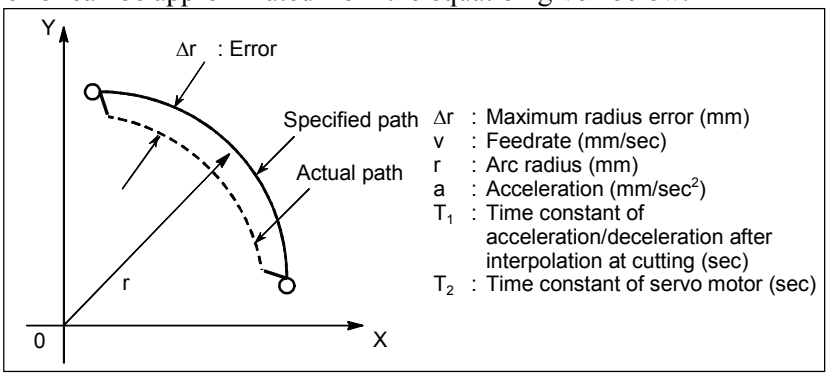


If "1" is set, the feedrate is determined not only with the condition that
the permissible feedrate difference and permissible acceleration on each axis are not exceeded, but also that the deceleration feedrate is constant regardless of the travel direction if the shape is the same.
If 1 is set for this parameter, the deceleration feedrate determined with the feedrate difference may be up to 30% lower than that determined if 0 is set.



- Speed control with acceleration in circular interpolation

When high-speed cutting is performed in circular interpolation, helical interpolation, or spiral interpolation, the actual tool path has an error with respect to the programmed path. In circular interpolation, this error can be approximated from the equation given below.



$$\Delta r = \frac{1}{2}(T_1^2 + T_2^2)\frac{v^2}{r} = \frac{1}{2}(T_1^2 + T_2^2)\cdot a \quad \text{.....(Equation 1)}$$

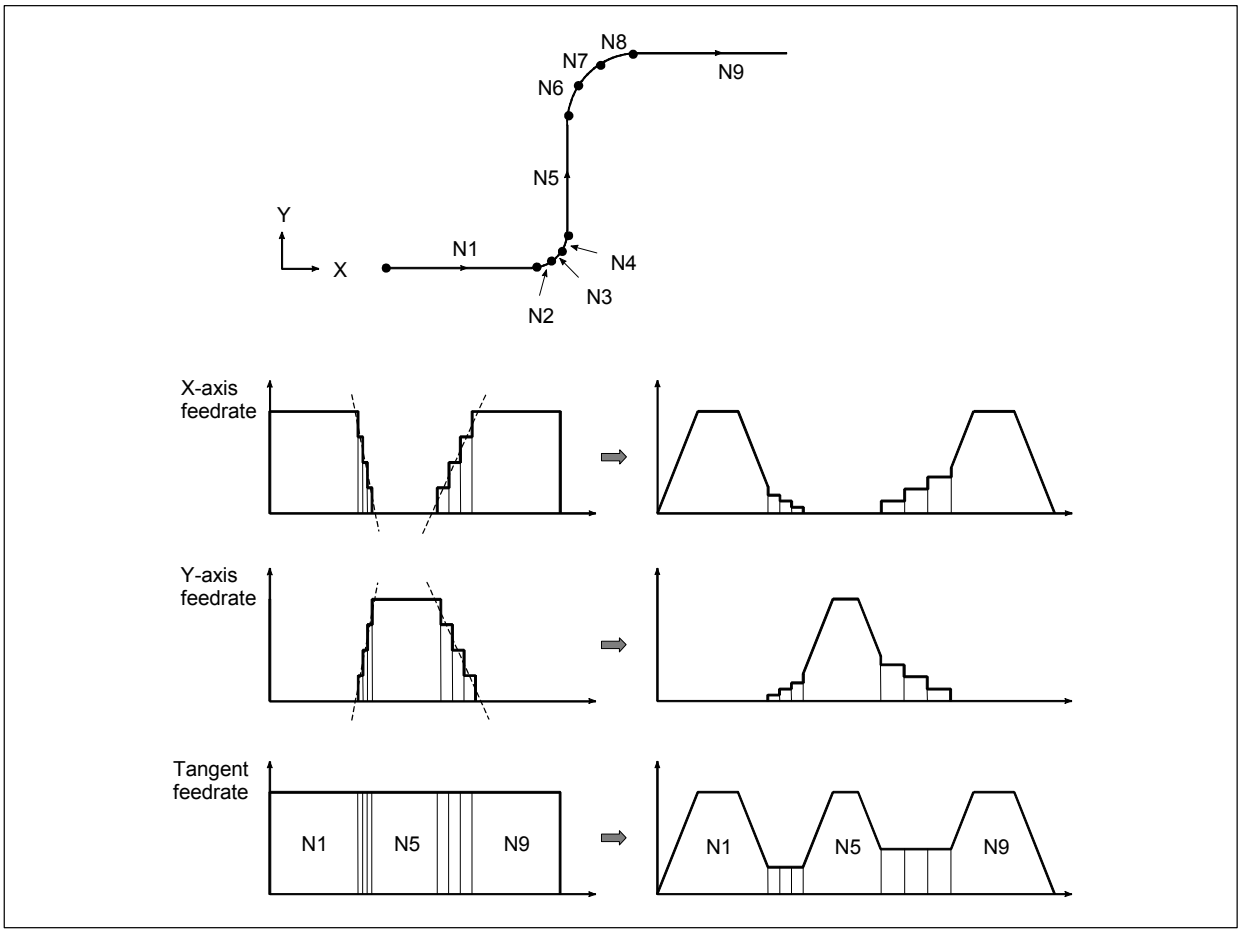
In actual machining, permissible error Δr is given, so the maximum permissible acceleration a (mm/sec^2) in equation 1 is determined.
When a specified feedrate causes the radial error from an arc having a programmed radius to exceed the permissible error, speed control with acceleration in circular interpolation automatically clamps the arc-cutting feedrate by using parameter settings.
Let the permissible acceleration calculated from the permissible acceleration set for each axis be A. Then, maximum permissible feedrate v with programmed radius r is expressed as follows:

$v = \sqrt{A \cdot r}$ .....(Equation 2)

If a specified feedrate exceeds feedrate v obtained from equation 2, the feedrate is clamped at feedrate v automatically.
The permissible acceleration is specified in parameter No. 1735. If there is a difference in permissible acceleration between two axes for circular interpolation, the lower acceleration is regarded as the permissible acceleration.
If the radius of an arc is small, too small value can be calculated as deceleration v. In such a case, the lower feedrate limit can be set in parameter No. 1732 to prevent the feedrate from being decreased too much.

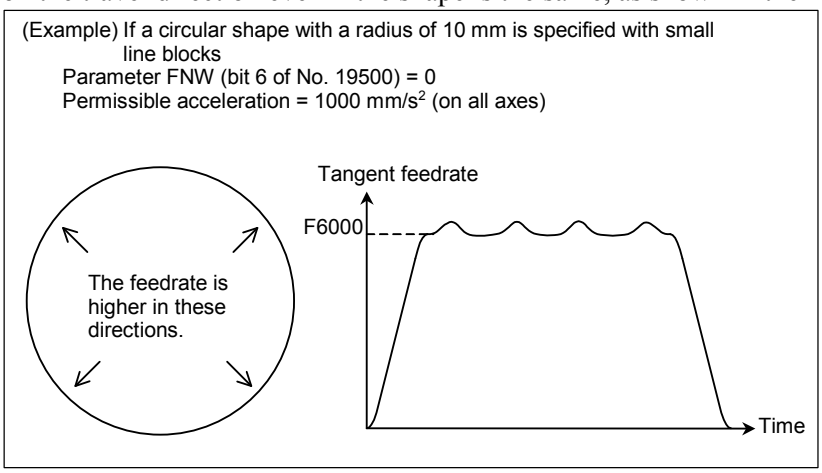
- Speed control with the acceleration on each axis

When consecutive small lines are used to form a curve, as in the example shown in the figure below, the feedrate differences on each axis at the individual corners are not very large. Thus, deceleration with the feedrate differences is not effective. Consecutive small feedrate differences, however, cause a large acceleration on each axis, as a whole.
In such a case, deceleration can be performed to reduce the impact on the machine and the machining error caused by too large an acceleration. The deceleration feedrate is determined to be the feedrate that does not cause the acceleration on each axis to exceed the permissible acceleration set for parameter No. 1737.
The deceleration feedrate is determined for each corner. The actual feedrate is the smaller of the deceleration feedrate determined at the start point of the block and that determined at the end point.
Depending on the specified figure, a very low deceleration feedrate may be calculated. In such a case, the lower feedrate limit can be set in parameter No. 1738 to prevent the feedrate from being decreased too much.
In the following example, the acceleration (gradient of the broken line in the feedrate graph) at too large at corners N2 to N4 and N6 to N8 and, therefore, deceleration is performed.



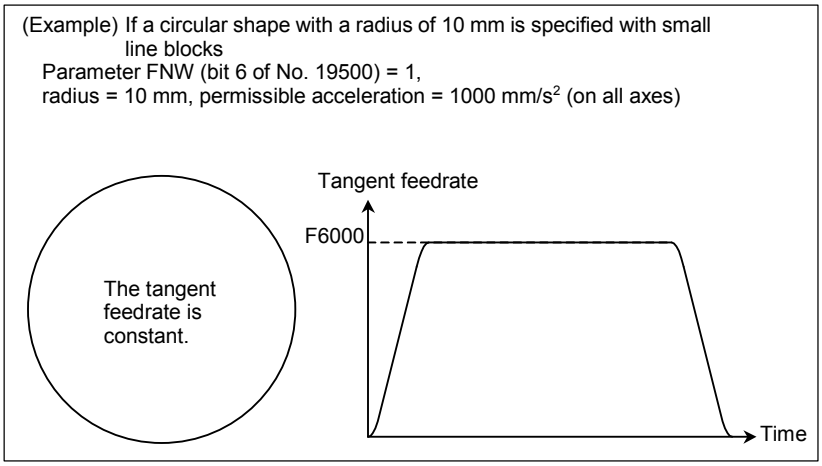
The method of determining the feedrate with the acceleration differs depending on the setting of bit 6 (FNW) of parameter No. 19500.

If "0" is set, the highest feedrate that does not cause the permissible acceleration set for parameter No. 1737 to be exceeded is assumed to be the deceleration feedrate. In this case, the deceleration feedrate differs depending on the travel direction even if the shape is the same, as shown in the figure below.



If "1" is set, the feedrate is determined with not only the condition that the permissible acceleration on each axis is not exceeded but also the condition that the deceleration feedrate is constant regardless of the travel direction if the shape is the same.

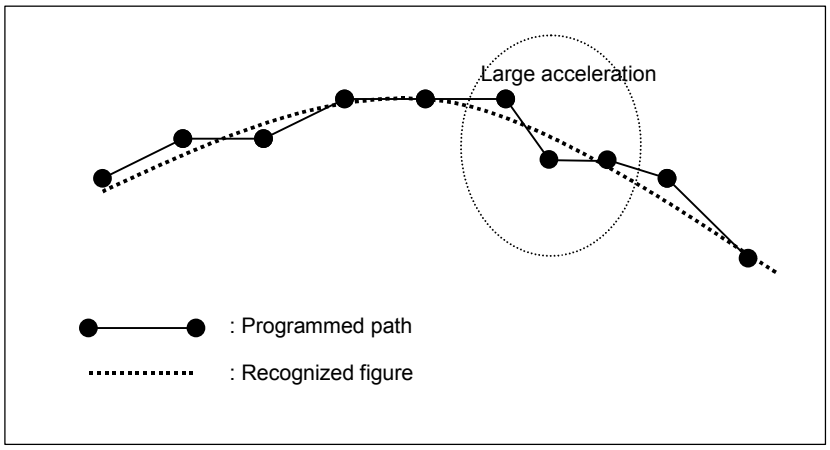
If 1 is set for this parameter, the deceleration feedrate determined with the feedrate difference or acceleration may be up to 30% lower than that determined if 0 is set.



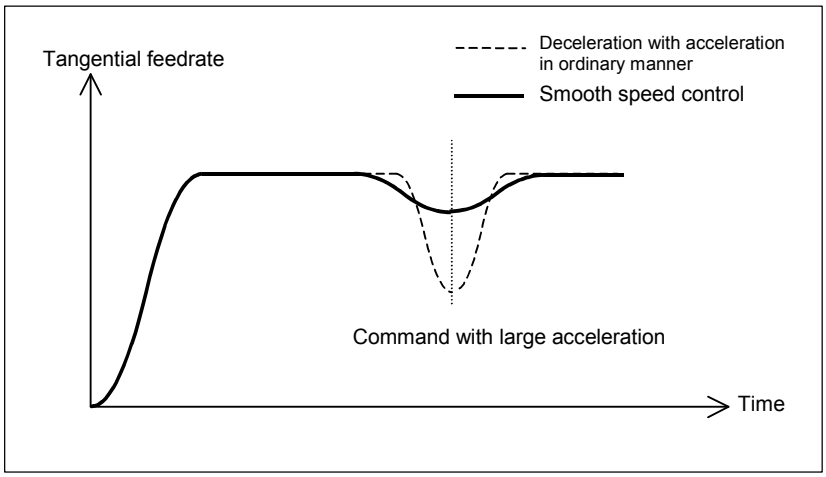
NOTE
In circular interpolation, the tangent feedrate is constant regardless of the setting of the parameter.

- Smooth speed control

In speed control with acceleration, the smooth speed control function recognizes the entire figure from preceding and following blocks including blocks read ahead to make a smooth feedrate determination.
When a curve is specified with successive minute straight lines, programmed values are rounded to the least input increment before issued, so the machining profile is approximated with a broken line.
When the feedrate is determined with acceleration in an ordinary manner, an optimum feedrate is automatically calculated exactly for a programmed figure, so a large acceleration may result depending on the command, which can lead to deceleration.
In such a case, the use of smooth speed control enables speed control by recognizing the entire figure, which provides smooth speed control while suppressing local deceleration, therefore increasing the feedrate.



Also for a part of a programmed figure in which a large acceleration would be required, the acceleration is obtained based on the figure recognized from multiple blocks, and the feedrate is determined so that the acceleration is within the permissible acceleration set in parameter No. 1737.

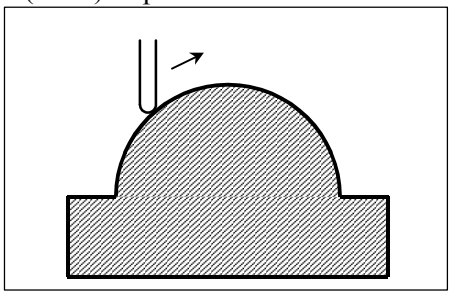


Smooth speed control obtains the acceleration by using the figure recognized from the preceding and following blocks including blocks read ahead, so smooth speed control is enabled even in parts in which the acceleration increases.
Smooth speed control is enabled under the following conditions:
<1> Speed control with acceleration is enabled in the AI contour control mode.
<2> Successive linear interpolation commands are specified.
<3> Bit 0 (HPF) of parameter No. 19503 is set to 1.

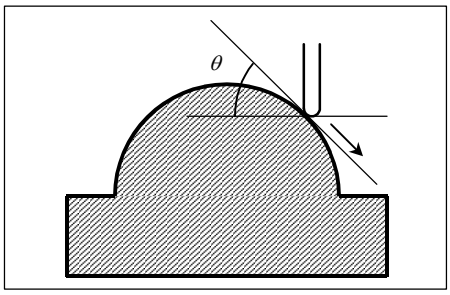
⚠ CAUTION
When smooth speed control is used, the feedrate in a certain figure such as a corner may become larger than the feedrate obtained by ordinary speed control with acceleration. For corners, set parameter No. 1783, which is the permissible feedrate difference parameter for speed control with the feedrate difference at corners, to perform appropriate deceleration by speed control with the corner feedrate difference.

- Speed control with the cutting load

Usually, the cutting resistance produced when machining is performed with the bottom of the cutter as the tool lowers along the Z-axis is greater than the cutting resistance produced when machining is performed with the side of the cutter as the tool rises along the Z-axis. Therefore, deceleration is required.
In AI contour control, the tool travel direction on the Z-axis is used as a condition for calculating the machining feedrate.
This function is enabled when bit 4 (ZAG) of parameter No. 8451 is set to 1.



During ascent on the Z-axis



During descent on the Z-axis

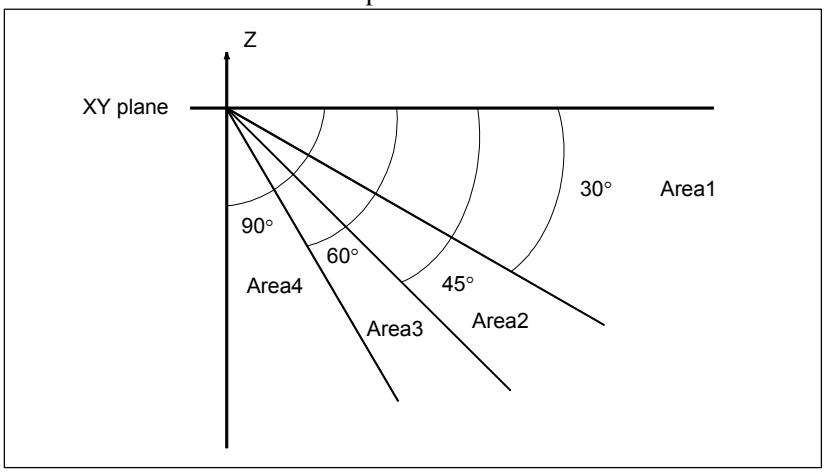
The descent angle θ during descent on the Z-axis (angle formed by the XY plane and the tool center path) is as shown in the figure. The descent angle is divided into four areas, and the override values for the individual areas are set for the following parameters:

Parameter No. 8456 for area 2
Parameter No. 8457 for area 3
Parameter No. 8458 for area 4

For area 1, however, no parameter is available, and an override of 100% is used at all times. The feedrate obtained according to other feedrate control is multiplied by the override value of the area to which descent angle θ belongs.

Area1 $0^\circ \leq \theta < 30^\circ$
Area2 $30^\circ \leq \theta < 45^\circ$
Area3 $45^\circ \leq \theta < 60^\circ$
Area4 $60^\circ \leq \theta < 90^\circ$

The feedrate can be overridden with an inclination by setting bit 1 (ZG2) of parameter No. 19515 to 1. In this case, specify the override value for area 1 in parameter No. 19516.



⚠ CAUTION

1 The speed control with the cutting feed is effective only when the tool is parallel with the Z-axis. Thus, it may not be possible to apply this function, depending on the structure of the machine used.
2 In the speed control with the cutting feed, the travel direction on the Z-axis is determined with the appropriate NC command. If, therefore, manual intervention is performed on the Z-axis with manual absolute on, or if a mirror image is applied on the Z-axis, the direction on the Z-axis cannot be determined. When using the speed control with the cutting load, do not use these functions.
3 When performing 3-dimensional coordinate conversion, determine the descent angle on the Z-axis using the converted coordinate system.

> **⚠ CAUTION**
> 4 Speed control with the cutting load is enabled for all interpolations in the AI contour control mode. This function, however, can be made valid only for linear interpolations by setting bit 4 (ZOL) of parameter No. 19503 to 1.

- Ignoring feedrate commands

In a block in which AI contour control is enabled, all feedrate commands (F commands) can be ignored by setting bit 7 (NOF) of parameter No. 8451.
The term feedrate commands, as used here, refer to the following commands:

<1> Modal F commands before the block in which AI contour control is enabled
<2> F commands and modal F commands in the block in which AI contour control is enabled

When the feedrate commands are ignored, it is assumed that the upper feedrate limit specified for parameter No. 8465 is specified.
Note, however, that any issued F commands and modal F commands are stored within the CNC.
Thus, in a block in which AI contour control changes from the enabled state to the disabled state, the modal values of the F commands described in <1> and <2> described above are used as modal F commands, instead of the modal values of the F commands calculated by AI contour control.

- Another example of determining the feedrate

If a specified feedrate exceeds the upper feedrate limit of AI contour control (in parameter No. 8465), the feedrate is clamped at the upper feedrate. The upper feedrate limit is clamped at the maximum cutting feedrate (parameter No. 1432).

Limitation

- Conditions for temporarily canceling the AI contour control mode

If one of the commands listed below is issued in the AI contour control mode, the AI contour control mode is canceled temporarily.
If the system becomes ready for AI contour control after it is canceled, the AI contour control mode is restored automatically.

- Positioning (rapid traverse)
- Single direction positioning
- Spindle positioning
- Rigid tapping
- Hypothetical axis interpolation
- Threading (single type, combined type)
- Electronic gear box
- Superimposed control
- When no move command is specified
- One-shot G code other than the following:
  - Tool offset
  - Cutter compensation vector retention
  - Cutter compensation corner rounding
  - Exact stop

Example

```
O0010
…
G5.1 Q1;
G01                    } AI contour control mode
…
X1.Y2.Z3.;
M220;   Start superimposed control
…                      } AI contour control mode is temporarily canceled.
M221;   Cancel superimposed control
X2.Y2.Z4.;             } Automatically return the system to AI contour control mode
…                      } AI contour control mode
X4.Y1.Z2.;
G5.1 Q0;
…
M30;
```

(**Note** The way to specify synchronous, composite, and superimposed controls differ from one machine tool builder to another. For details, refer to the manual issued by the machine tool builder.)

- Functions that cannot be specified in the AI contour control mode

In the AI contour control mode, the functions listed below cannot be specified. Before specifying these functions, turn off the AI contour control mode; after the command ends, turn on the mode again.

- Threading
- Circular threading
- Variable lead threading

Threading, circular threading, and variable lead threading can be specified in the AI contour control mode by setting bit 1 (THA) of parameter No. 1611. However, the AI contour control mode is automatically canceled.

- Modal G codes usable when AI contour control is specified

In a modal G code state listed below, AI contour control can be specified.
Do not specify AI contour control in modal states other than these.

G13.1 : Polar coordinate interpolation mode cancel
G15 : Polar coordinate command cancel
G40 : Tool radius compensation cancel
G40.1 : Normal direction control cancel
G50 : Scaling cancel
G50.1 : Programmable mirror image cancel
G50.2 : Polygon turning cancel
G54.2P0 : Rotary table dynamic fixture offset cancel
G54.4P0 : Workpiece setting error compensation cancel
G64 : Cutting mode cancel
G67 : Macro modal call cancel
G69 : Coordinate system rotation/3-dimensional coordinate system conversion cancel
G80 : Canned cycle cancel
G97 : Constant surface speed control cancel

Notes

- About processing macro statements

In AI contour control mode, the NC statements of multiple blocks are looked ahead. Macro statements such as arithmetic expressions and conditional branches are processed as soon as they are read into the buffer. Therefore, the timing of the macro statement execution is not always the specified order.
In case that you need to execute the macro statement after completing the NC block just before the macro statement, specify M code or G code that is not buffered just before the macro statement. Specially, in case of reading/writing the system variables to control signals, coordinates, offset value, etc., it may be different system variable data by the timing of the NC statement execution. To avoid this phenomenon, specify such M codes or G codes before the macro statement, if necessary.

20.2 MACHINING CONDITION SELECTING FUNCTION

Overview

By setting a speed- or precision-focused parameter set in an AI contour control function and specifying a precision level in accordance with the machining conditions during machining, parameters suitable to the conditions can be automatically calculated so that machining can be performed.
This function is an optional one.

Format

- Changing the precision level using a program

In addition to being switched on the precision level selection screen, the precision level can be changed using a program in the format below.

```
G05.1 Q1 Rx ;
  x.....Level (1 to 10)
```

The precision level can also be changed in the format used with conventional "advanced preview control", "high precision contour control", and "AI high precision contour control".

```
G05 P10000 Rx ;
  x.....Level (1 to 10)
G08 P1 Rx ;
  x.....Level (1 to 10)
```

⚠ CAUTION
Once specified, a level remains effective even if the AI contour control mode is canceled.

20.3 MACHINING QUALITY LEVEL ADJUSTMENT

Overview

In nano smoothing, if the “level 1” and “level 10” parameters of a precision level and smoothing level are set in order to specify a precision level and smoothing level according to the machining condition during machining, the parameter values corresponding to the condition can be automatically calculated for machining.
On the machining quality level adjustment screen, the machining quality/precision/speed level in nano smoothing can easily be adjusted.
This function is an optional function.

Format

- Changing the smoothing level by a program

The smoothing level can be switched on the machining level selection screen or machining quality level adjustment screen; it can also be changed by a program with the following format.

G05.1 Q3 Rx ;

x .....Level (1 to 10)

⚠ CAUTION

Once a level is specified, it remains valid even after the nano smoothing mode is canceled.

- Changing the precision level by a program

For information on the changing the precision level by a program, see Section 20.2, "MACHINING CONDITION SELECTION FUNCTION".

20.4 HIGH-SPEED CYCLE MACHINING

This function converts the shape to be machined into a data group that can be subject to high-speed pulse distribution, using the macro executor, calls the data group with a CNC command (G05 command), and executes it.

Format

G05 P10xxx Lxxx ;

P10001 to P10999 : Start number of the machining cycle to call
L1 to L999 : Repeat count of the machining cycle (assumed to be L1 if omitted).

Cycle data for up to 999 cycles can be created. Use address "P" to specify which cycle to use for machining. It is possible to call multiple cycles in succession using connection information (in a header) and execute them.
Use address "L" to specify the repeat count of the machining cycle. It is possible to specify the repeat count (in a header) for each cycle.
Connection of cycles and the repeat count are explained with the following example.

Example)
If
Cycle 1 Connection information 2 Repeat count 1
Cycle 2 Connection information 3 Repeat count 3
Cycle 3 Connection information 0 Repeat count 1
G05 P10001 L2;
the following cycles are executed in succession:
1,2,2,2,3,1,2,2,2,3

NOTE
1 If an attempt is made to issue the function in the modes below, an alarm is issued.
- Hypothetical axis interpolation (G07)
- Cylindrical interpolation (G07.1)
- Polar coordinate interpolation (G12.1)
- Polar coordinate command (G16)
- Spindle speed fluctuation detection (G26)
- Tool radius·Tool nose radius compensation (G41/G42)
- Normal direction control (G41.1/G42.1)
- Scaling (G51)
- Programmable mirror image (G51.1)
- Coordinate system rotation (G68)
- Canned cycle (G81 to G89)
- Constant surface speed control (G96)
- Macro interrupt (M96)

2 During high-speed cycle machining, single block stop is disabled.

20.5 HIGH-SPEED BINARY PROGRAM OPERATION

High-speed binary program operation creates a profile to be machined as a data group that can be processed with high-speed pulse distribution in an external program and executes the external program by DNC operation or external subprogram call (M198).

Format

- Pulse distribution period

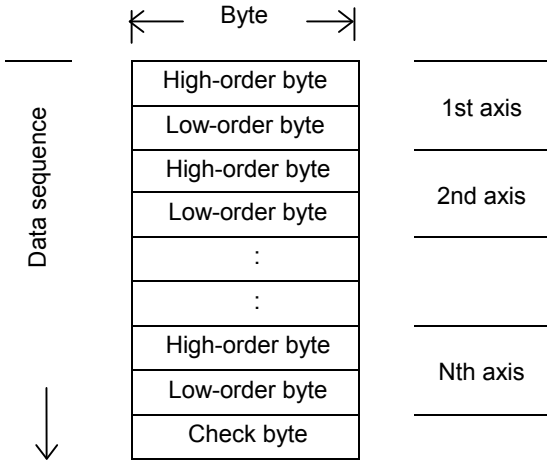
Set the number of pulses in the command format. The unit for the number of pulses is the least input increment.

- Command format

Register a program in the command format in an external device to execute the program by DNC operation or external subprogram call operation.
For operation of data of the profile to be machined, specify "G05;" only in a block in the normal NC command format and move data in the format shown below. The normal NC command format is used again for the subsequent command specification by specifying the travel distance along all axes to zero.

- High-speed binary program operation on : G05;
- High-speed binary program operation off : The travel distances along all axes are set to zero.

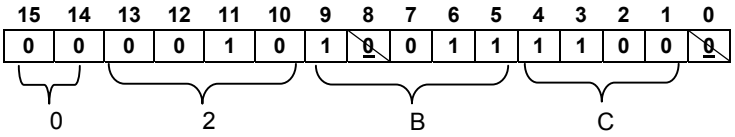
Data format for high-speed binary program operation



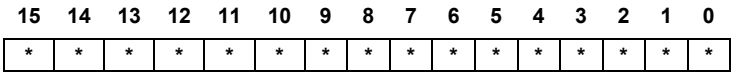
1) The travel distance along each axis (2 bytes) per unit time is placed sequentially for all axes from the 1st axis, followed by a check byte (1 byte). (The data length for one block is (2 × N + 1) bytes.)
2) Data must be all represented in binary. (A negative travel distance is represented in 2's complement notation.)
3) The data format of a travel distance is shown below:
   - When the bit 3 (BDS) of parameter No.7501 is set to 0:
     A travel distance per unit time is specified using the bits marked with an asterisk (*).

15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
*	*	*	*	*	*	*	**0**	*	*	*	*	*	*	*	**0**

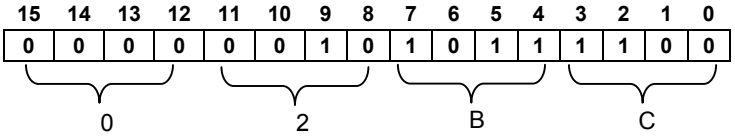
Example) When the travel distance is 700 μm per unit time (millimeter machine with increment system IS-B)
Since 700 in decimal equals 02BC in hexadecimal, the travel distance is specified as follows:



- When the bit 3 (BDS) of parameter No.7501 is set to 1:
A travel distance per unit time is specified using the bits marked with an asterisk (*).



Example) When the travel distance is 700 μm per unit time (millimeter machine with increment system IS-B)
Since 700 in decimal equals 02BC in hexadecimal, the travel distance is specified as follows:



4) For the check byte, all bytes in the block other than the check byte ((2 × N) bytes) are summed up in bytes and any bits above the 8th bit are discarded.

20.6 OPTIMUM ACCELERATION/DECELERATION FOR RIGID TAPPING

Overview

This function can be used to flexibly set the acceleration/deceleration during cutting in rigid tapping according to the torque characteristics of a spindle motor and the mechanical characteristics such as machine friction. Depending on the torque characteristics of a spindle motor and the mechanical characteristics, the acceleration/deceleration performance (referred to below as the maximum acceleration curve) that can be output is not symmetrical in the low-speed and high-speed parts. The conventional acceleration/deceleration (linear-shaped/bell-shaped) is symmetrical, so it was impossible to make the most of the motor performance.
This function can be used to perform acceleration/deceleration so that the actual speed curve can follow the maximum acceleration curve as close as possible. This can make the most of the motor performance and reduce the cutting time. When rigid tapping is used in an area where the acceleration of the spindle motor is constant, however, reduction in the cutting time cannot be expected.
If the acceleration pattern is set for the parameter for each gear, rigid tapping by acceleration following the maximum acceleration curve.
In acceleration/deceleration during withdrawal, the acceleration/deceleration pattern during cutting is used.

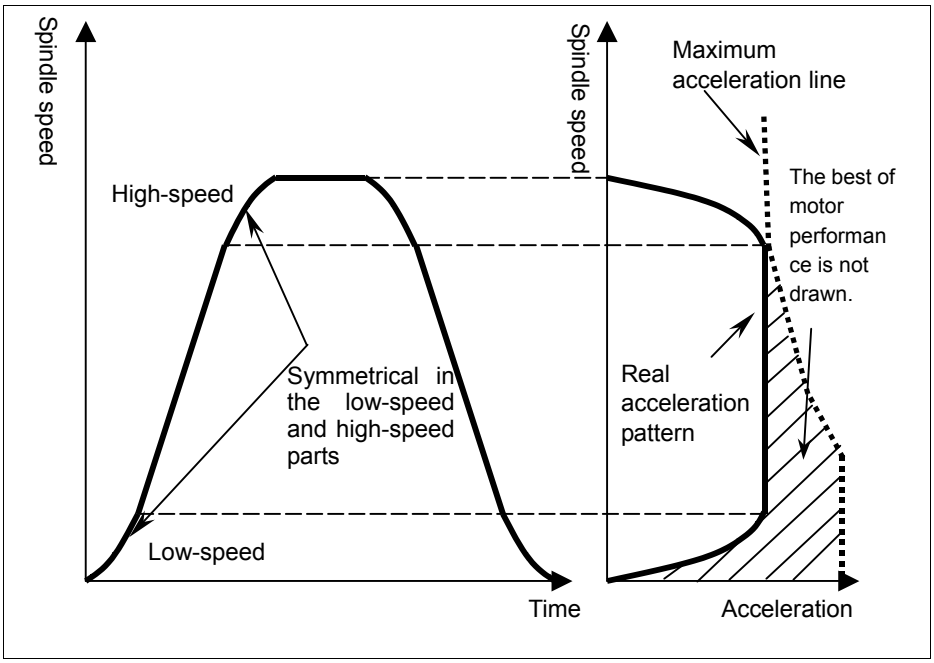


Fig. 20.6 (a) Conventional acceleration/deceleration (bell-shaped)

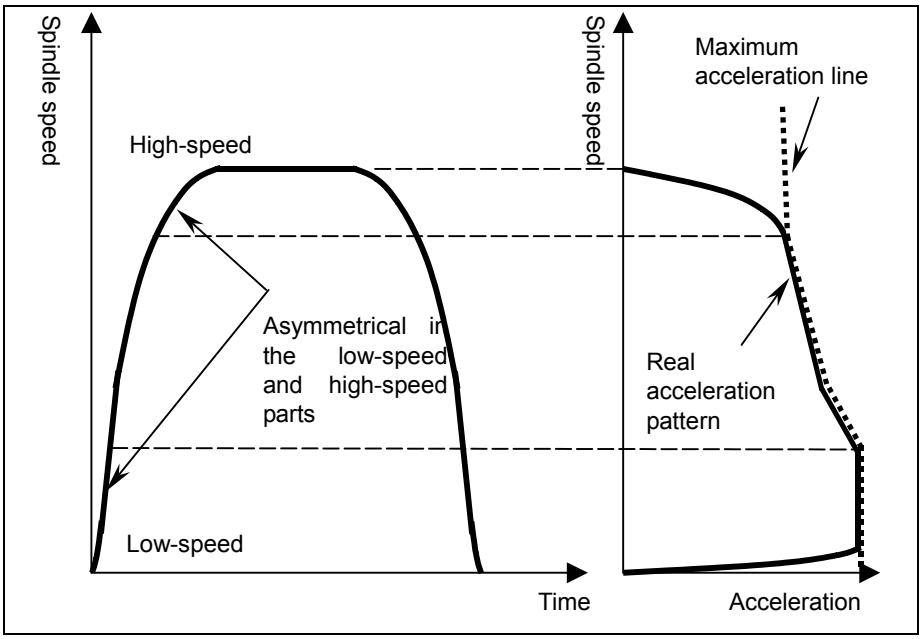


Fig. 20.6 (b) Acceleration/deceleration in which the maximum acceleration curve is followed by the actual acceleration curve by this function

20.7 PATH TABLE OPERATION

Overview

The Path Table Operation controls each axis independently, based on the Path Table of each axis memorized in the part program memory in synchronization with the time or the spindle/axis position.

The general data flow of the Path Table Operation is as follows.

(1) Before starting Path Table Operation, Path Tables memorized in the part program memory need to be converted to the executive form data.
(2) The reference value is calculated based on either the time with considering override, the spindle position or the axis position.
(3) The interpolation for each axis is carried out, by using the Path Table and the reference value respectively.

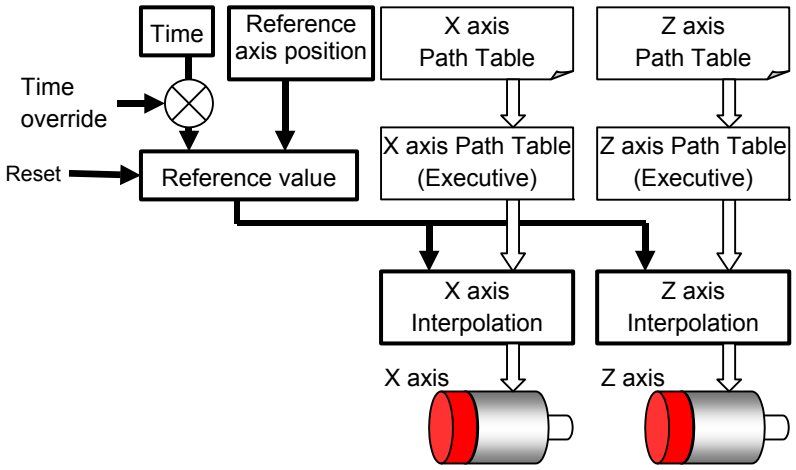


Fig. 20.7 (a) General data flow of Path Table Operation

Explanation

Command of Machine Coordinate System or Workpiece Coordinate System

If bit 0 (PCA) of parameter No. 11104 is 0, axis position and reference position in Path Table Operation are the value of machine coordinate system.
If bit 0 (PCA) of parameter No. 11104 is 1, axis position and reference position in Path Table Operation are the value of workpiece coordinate system.

Classification of Path Table

The Path Tables are classified as follows.

- Axis Motion Table
- Spindle Command Table (spindle speed command, spindle synchronization start/end, etc.)
- Auxiliary Function Table

Axis Movement Table defines movements of tool axis. One table defines movements of one tool axis.
Spindle Command Table defines spindle commands. One table defines commands for one spindle axis (or Cs contouring control axis or axes of live tool control with servo motor).
Auxiliary Function Table defines M codes to be output. One table defines M code for one path.

Configuration of Path Table

One Path Table consists of Program Name, Table Header, Table Data and Table Footer as follows.
The maximum number of characters in 1 block is 128 including comment.

```
%
<TIME_TABLE_66_X1>;     Program Name
R98;                    Table Header
L1000 X10.0;            Table Data
  :
L7000 X200.0;           Table Data
R99;                    Table Footer
%
```

Any comments can be inserted in the Path Table by sandwiching them with parentheses “(“ and “)”.

Program Folder of Path Table

The programs for the Path Tables are stored in the sub-folder of the part program files, which has the name "PATHTABLE/". "PATHTABLE/" folders must be placed under path(head) folders. All programs in the "PATHTABLE/" folder are converted to the executive form by one conversion management.
Example)

Path 1 //CNC_MEM/USER/PATH1/PATHTABLE/
Path 2 //CNC_MEM/USER/PATH2/PATHTABLE/

It is not allowed for the Path Table to command axes which belong to the other path(head). When synchronous or interpolated motion is required beyond path(head)s, the Path Table should be programmed for each path(head), and be started at the same time.

The Path Tables should be downloaded for each path(head) respectively.
C Language Executor can download the program, which is stored in the memory card in the slot on the LCD unit.
FOCAS2 library can download the program, which is stored in the PC.

Program Name

The maximum of the Path Table number is 9999. Path Tables are stored in the ordinary part program memory. So, the actual program numbers to be registered cannot exceed the maximum number to be registered in the part program memory.

- Axis Motion Table

The program name varies depending on whether the reference value is time or axis or spindle position. If the different kind of axis motion tables which have the same axis name and the same table number each other exists, the alarm is issued when the Path Table is called by M code for Path Table Operation.

- Time Reference
  In case that the Path Table is defined based on the time, the program name should be as follows.
  <TIME_TABLE_"Table Number(4-digit)"_"Axis Name">
  The unit of time is msec. The command can be specified in each 1msec.
  Example)
  <TIME_TABLE_0001_X1>, <TIME_TABLE_0012_Z1>
- Axis Position Reference
  In case that the Path Table is defined based on the axis, the program name should be as follows.
  <AXIS_TABLE_"Table Number(4-digit)"_"Axis Name">
  A reference axis is specified in the table header. For details, refer to “Table Header”.
  Example)
  <AXIS_TABLE_0001_Y2> (A table of axis to make axis position reference needs to be a time reference table.)

- Spindle Position Reference
  In case that the Path Table is defined based on the spindle position, the program name should be as follows. The spindle position reference is available for the threading.
  <SPDL_TABLE_"Table Number(4-digit)"_"Axis Name">
  A reference spindle is specified in the table header. For details, refer to “Table Header”.
  Example)
  <SPDL_TABLE_0101_X3>, <SPDL_TABLE_0101_Z3>

- Auxiliary Function Table

The program name varies depending on whether the reference value is time or axis or spindle position. If the different kind of auxiliary function tables which have the same table number each other exist, the alarm is issueed when the Path Table is called by M code for Path Table Operation.

- Time Reference
  In case that the Path Table is defined based on the time, the program name should be as follows.
  <TIME_TABLE_"Table Number(4-digit)"_M>
  The unit used in specifying time is msec. The command can be specified in each 1msec.
- Axis Position Reference
  In case that the Path Table is defined based on the axis position, the program name should be as follows.
  <AXIS_TABLE_"Table Number(4-digit)"_M>
  A reference axis is specified in the table header. For details, refer to “Table header”.
- Spindle Position Reference
  In case that the Path Table is defined based on the spindle position, the program name should be as follows.
  <SPDL_TABLE_"Table Number(4-digit)"_M>
  A reference spindle is specified in the table header. For details, refer to “Table header”.

- Spindle Command Table

Only the time reference is selected in case of the spindle command table. The axis position reference and the spindle position reference are not available for spindle command table. The program name should be as follows.

<TIME_TABLE_"Table Number(4-digit)"_"Spindle Name">

The spindle name is specified with the extended spindle name. As to the extended spindle name, refer to “EXTENDED SPINDLE NAME” in this manual.

Example)

<TIME_TABLE_0001_S11>, <TIME_TABLE_0001_S2>

The axis controlled by the function of live tool control with servo motor should be commanded on spindle command table.

Table Header

At the head of each Path Table, the following command should be designated. One header can be commanded on one Path Table. One skip condition can be commanded on one Path Table.

- Format

(IP_=0) (S_=0) (I_) (Q_) (P_) (K_) R98 ;

IP_	:	Reference Axis Name
S_	:	Reference Spindle Name
I_	:	Spindle Position Corresponding to Reference L0
Q_	:	Path Table Number to be connected
P_	:	Path Table Number in case of Skip
K_	:	Path Table Skip Condition
R98	:	Indication of Path Table Header

(1) Reference Axis Name
In case that the Path Table is based on the axis position, the reference axis name is defined in IP code.
Example)
In case that axis X1 is the reference axis.
<AXIS_TABLE_0123_Y1> ;
X1=0 R98 ;

(2) Reference Spindle Name
In case that the Path Table is based on the spindle position, the reference spindle name is defined in S code.
Example)
In case that spindle S21 is the reference spindle.
<SPDL_TABLE_0101_Z1> ;
S21=0 R98 ;

(3) Spindle Position Corresponding to Reference L0
The spindle position, which corresponds to the reference L0, is specified in I address. For the details of L address, refer to “Data of Axis Motion Table”.
If bit 0 (PCA) of parameter No. 11104 = 0, axes start moving after the spindle position reaches the specified coordinate of machine coordinate system.
If bit 0 (PCA) of parameter No. 11104 = 1, axes start moving after the spindle position reaches the specified coordinate of workpiece coordinate system.

When I0 is commanded, axes start moving after the spindle position reaches 0 degrees.
When I90 is commanded, axes start moving after the spindle position reaches 90 degrees.
When I address is ignores, the current spindle position is regarded as L0. In other words, axes start moving immediately.
Example)

```
<SPDL_TABLE_0001_Z1>;
S21=0 I90.0 R98 ;     Wait till spindle S21 reaches 90 degrees.
:
L45 X60.0 ;           Position of axis Z1 is 60.0mm
:                     when spindle S21 reaches 90+45=135 degrees.
                      (For details of L_X_ command, refer to “Data of Axis Motion Table”.)
```

(4) Path Table Number to be connected
The next Path Table number, which is connected to the current Path Table, is specified in Q address. After the execution of the current Path Table, the next Path Table continues automatically. When there is no Q address designation, it is regarded that the Path Table Operation is terminated at the table. The reference value is common during the connection of the Path Tables. The reference value should not be changed.
It is not necessary to command the Path Table number in order. The connected Path Table is executed without delay. To execute the loop operation repeatedly, it is necessary to return to ISO code and call the same Path Table again.
And before and after the Path Table connection, the kinds of reference have to be same. From the Path Table of time reference, it can be connected only to the Path Table of time reference. And from the Path Table of spindle position reference, it can be connected only to the Path Table of spindle position reference.

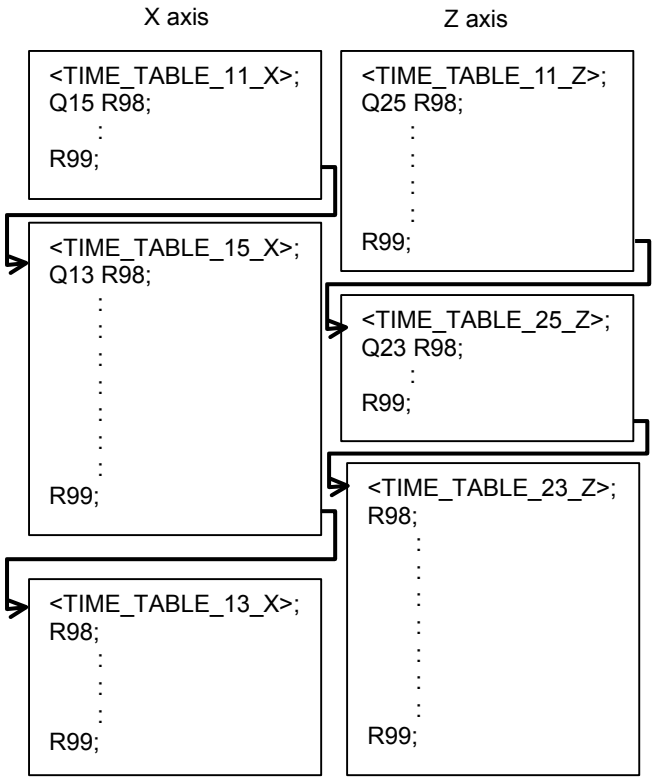


Fig. 20.7 (b) Path Table connection

(5) Path Table Skip Condition

The Path Table skip condition is specified in K address, and the Path Table number in case of skip is specified in P address. At the start of each Path Table, Path Table skip signals PTSK1 to PTSK8 are checked. The signal status is coincident with the specified condition, the Path Table execution is skipped, and the Path Table specified in P address is executed instead.
One skip condition can be commanded on one Path Table.

K1 : Skip by Path Table Skip Signal PTSK1.
K2 : Skip by Path Table Skip Signal PTSK2.
K3 : Skip by Path Table Skip Signal PTSK3.
K4 : Skip by Path Table Skip Signal PTSK4.
K5 : Skip by Path Table Skip Signal PTSK5.
K6 : Skip by Path Table Skip Signal PTSK6.
K7 : Skip by Path Table Skip Signal PTSK7.
K8 : Skip by Path Table Skip Signal PTSK8.

(6) Path Table Number in case of Skip

The Path Table number to move when it skips is specified in the P address. When the Path Table number isn't specified in the P address, it moves to the Path Table specified in the connection of the Path Table number (Q address) when skip condition corresponds.
The Path Table Skip Signal is effective only at the header of each table.

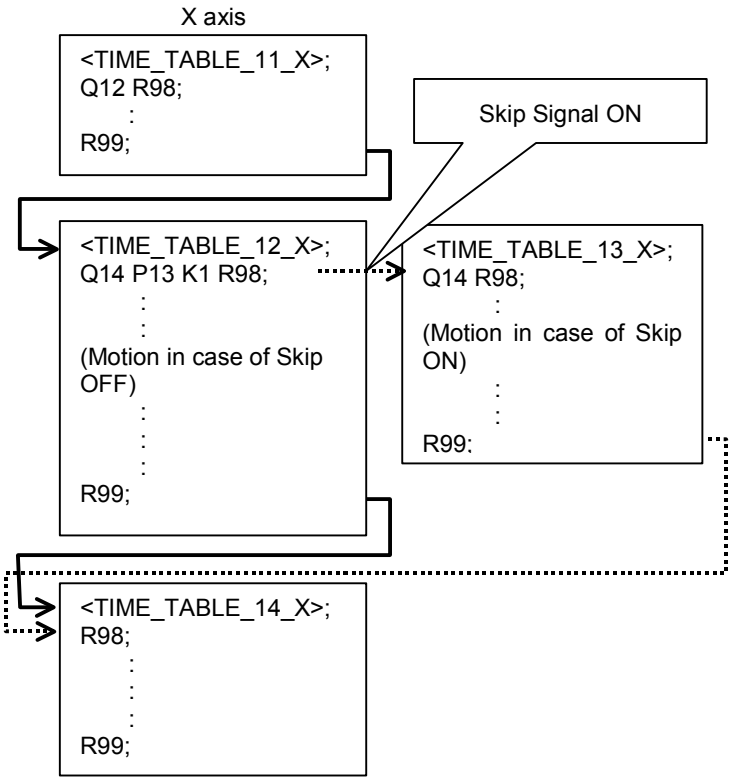


Fig. 20.7 (c) Path Table Skip

(7) Indication of Path Table header
"R98" is specified in order to indicate Table header definition.

Data of Axis Motion Table

Data of Axis Motion Table memorizes the each axis position corresponding to the reference values such as the time or the axis position or the spindle position. The data is configured as follows.

Table 20.7 (a) Reference values and axis positions of Axis Motion Table

Reference value	Axis position
L0	X0
L1	X1
L2	X2
L3	X3
L4	X4

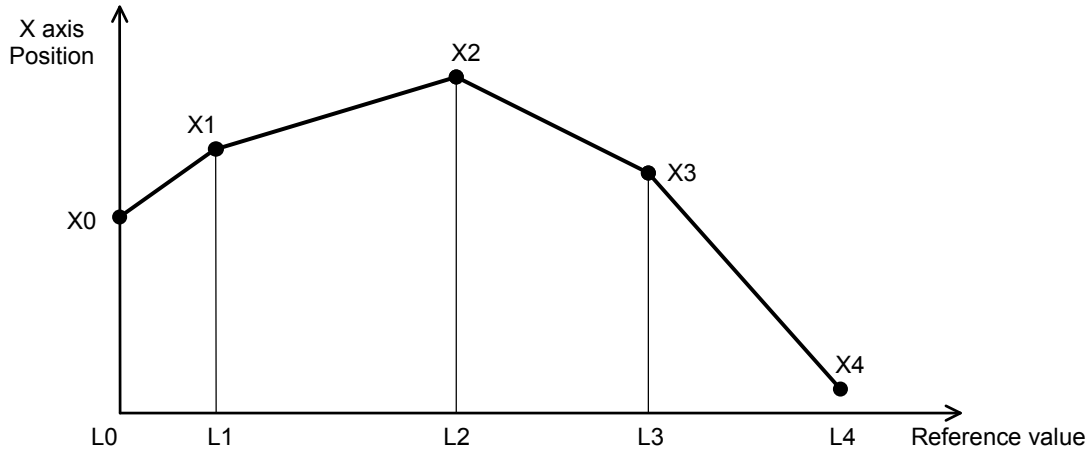


Fig. 20.7 (d) Reference values and axis positions of Axis Motion Table

- Format

```
L_ X_ ;
  L_    :   Reference Value
  X_    :   Axis Position (mm / inch / degree), Radius Value
```

(1) Reference Value

The reference value is designated in L address. The value should increase gradually. The range of designation is as follows. Time Reference and the axis position reference and the spindle position reference should be commanded with the absolute value.

Time Reference: L0 to L99999999.999999
(a unit is msec, it is specified in the multiple of 1. A decimal point can be specified, but it is rounded off.)
Axis Position Reference: L-999999999 to L999999999
(Unit is least input increment.)
If bit 0 (PCA) of parameter No. 11104 = 0, coordinate of machine coordinate system is commanded.
If bit 0 (PCA) of parameter No. 11104 = 1, coordinate of workpiece coordinate system is command.
Spindle Position Reference: L-999999999999.999 to 999999999999.999 degree
The reference value should increase gradually.

(2) Axis Position

The axis position corresponding to the reference value is specified in X address.
If bit 0 (PCA) of parameter No. 11104 = 0, coordinate of machine coordinate system is commanded. The unit of the coordinate is not the unit of the machine coordinate but the unit of the workpiece coordinate.
If bit 0 (PCA) of parameter No. 11104 = 1, coordinate of workpiece coordinate system is command. The unit of the coordinate is not the unit of the machine coordinate but the unit of the workpiece coordinate.
Even a diameter designated axis always commands by the radius designation.

Table 20.7 (b) Range of Axis Position Specifications

Least Input Increment	Valid Data Range
IS-B	-999999.999 mm to 999999.999 mm -99999.9999 inch to 99999.9999 inch -999999.999 deg to 999999.999 deg

Least Input Increment	Valid Data Range
IS-C	-99999.9999 mm to 99999.9999 mm -9999.99999 inch to 9999.99999 inch -99999.9999 deg to 99999.9999 deg

The axis positions between two specified axis positions are interpolated linearly. The feedrate is automatically specified from the commanded position.

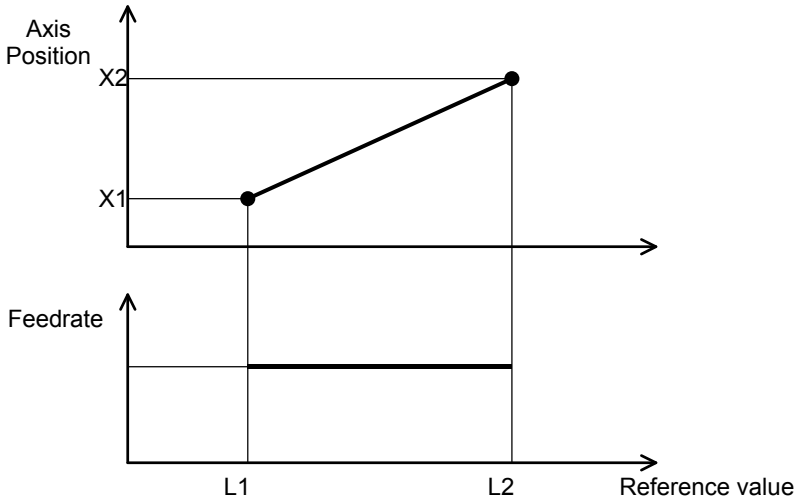


Fig. 20.7 (e) Position and feedrate when axis motion commands are specified

Data of Spindle Command Table

- Format

Spindle Speed Control

L_ S_ (R11) ;

L : Reference Value
S : Spindle speed (min^{-1})
R11 : Definition of Spindle Speed Control. R11 can be omitted.

Start of Spindle Synchronization

L_ S_=0 (I_) R21 ;

L_ : Reference Value
S_ : Name of Master Spindle
I_ : Phase of Synchronization (degrees) I address can be omitted.
R21 : Definition of Start of Spindle Synchronization

End of Spindle Synchronization

L_ R20 ;

L_ : Reference Value
R20 : Definition of End of Spindle Synchronization

Start of Contouring Control

L_ X_ R31 ;

L_ : Reference Value
X_ : Coordinate of Axis (degrees)
R31 : Definition of Start of Contouring Control

In Contouring Control

L_ X_ ;

L_ : Reference Value
X_ : Coordinate of axis (degrees)

End of Contouring Control

L_ R30 ;

L_ : Reference Value
R30 : Definition of End of Contouring Control

Start of Constant Surface Speed Control

L_ IP=_ S_ R41 ;

L_ : Reference Value
IP_ : Standard Axis and Coordinate of Constant Surface Speed Control
S_ : Speed at Surface
R41 : Definition of Start of Constant Surface Speed Control

Clamp Speed of Constant Surface Speed Control

L_ S_ R42 ;

L_ : Reference Value
S_ : Maximum clamp speed in Constant Surface Speed Control (min^{-1})
R42 : Definition of Clamp Speed of Constant Surface Speed Control

End of Constant Surface Speed Control

L_ R40 ;

L_ : Reference Value
R 40 : Definition of End of Constant Surface Speed Control

NOTE
Make the spindle Cs contour control mode to execute the spindle command table in the Path Table Operation. For details, Refer to “Cs CONTOUR CONTROL” in this manual.

(1) Reference Value
The reference of the time is specified in L address. The value should increase gradually. The range of designation is as follows. Time Reference should be commanded with the absolute value.

Time Reference: L0 to L99999999.999999
(A unit is msec, it is specified in the multiple of 1. A decimal point can be specified, but it is rounded off.)

(2) Definition R
The definition of the spindle command is specified in R address as follows.

Table 20.7 (c) Definition R

Definition	Function
R11	Spindle Speed Control
R21	Start of Spindle Synchronization
R20	End of Spindle Synchronization
R31	Start of Contouring Control
R30	End of Contouring Control
R41	Start of Constant Surface Speed Control
R40	End of Constant Surface Speed Control
R42	Clamp Speed of Constant Surface Speed Control

(3) Spindle Speed Control L_ S_ (R11) ;
The spindle speed corresponding to the reference time (L_) is specified in S address by the unit of min^{-1}. The spindle is accelerated for the commanded speed when spindle speed control is commanded. R11 can be omitted. The valid range is from -99999 to 99999 min^{-1}. The spindle stops at the transit between spindle speed command and spindle contouring mode.
Example)

```
%
<TIME_TABLE_0012_S11> ;   Path Table (Time Reference) 0012 for S11
Q0013 R98;                Next Path Table is 13
L0 S0 R11 ;               Spindle Speed Control
L1000 S1000 ;             1000 min-1 (Clockwise)
L2000 S800 ;              800 min-1 (Clockwise)
L3000 S-2000 ;            2000 min-1 (Counter-clockwise)
:
R99 ;
%
```

The operation of R11 command in the modes other than spindle speed control is as follows:
In Spindle Contouring Control Mode (R31) :
Spindle mode is forced to be the spindle speed control mode. It is equivalent to G96.4 S_ P_ ;.
In Spindle Synchronization (R21) :
The spindle speed at the end of synchronization (R20) is commanded. The spindle synchronous state continues.
In Constant Surface Speed Control (R41):
The constant surface speed is updated with the value of S address. The constant surface speed control continues.

(4) Spindle synchronization
Spindle synchronization command is specified in the Path Table of the slaved spindle. In order to activate the spindle synchronization, bit 4 (SSS) of parameter No. 3704 should be set to 1. It makes the individual spindle synchronization effective. The Path Table Operation requires the extended spindle name. One master spindle can have 7 slave spindles in Path Table Operation.

- Start of Spindle Synchronization L_ S_=0 (I_) R21 ;
  If bit 0 (PCA) of parameter No. 11104 is 0, the phase difference (I_) is the difference of machine coordinate system.
  If bit 0 (PCA) of parameter No. 11104 is 1, the phase difference (I_) is the difference of workpiece coordinate system.
  The slave spindle starts synchronizing with the master spindle (S_=0) at the reference time (L_).
  The phase difference is specified in I address.
  The extended spindle name is used. The master spindle name is specified in S address. For details of extended spindle name, refer to "EXTENDED SPINDLE NAME" in this manual.
  The command is ignored if the spindles are already in the spindle synchronization mode (R21).
  Example)
  At the reference time 100, S21 starts synchronizing with master S11 with phase difference 90.0 degrees (machine coordinate).
  <TIME_TABLE_0001_S21>; Spindle Command Table for S21
  :
  L100 S11=0 I90.0 R21 ;
  :

- End of Spindle Synchronization L_ R20 ;
  Spindle synchronization is cancelled at the time specified L address. The spindle returns to the speed control mode, and rotates by the previously specified speed. The spindle speed becomes the value of the last spindle speed Command (R11).
  Example)
  At the time 200, the spindle synchronization is cancelled.
  <TIME_TABLE_0001_S21>; Spindle Command Table for S21
  :
  L200 R20 ;
  :

(5) Contouring Control

- Start of Contouring Control L_ X_ R31 ;
  The spindle mode is changed to the contouring control mode at the reference time (L_), and the spindle is decelerated and stopped at the position (X_). Shortcut control is not executed. The command is ignored when the spindle is already in contouring control mode (R31).
  If bit 0 (PCA) of parameter No. 11104 is 0, the stop position (X_) is the value of machine coordinate system.
  If bit 0 (PCA) of parameter No. 11104 is 1, the stop position X_ is the value of workpiece coordinate system.

- In Contouring Control L_ X_ ;
  During contouring control, the data is the same as the data of the axis motion table. Refer to “Data of Axis motion table”.

- End of Contouring Control L_ R30 ;
  The spindle is released from contouring control at the reference time (L_), and stops after returning to the speed control mode. The spindle does not stop when the spindle is already in the spindle speed control mode (R11).

NOTE
1 Spindle Synchronization is not executed in contouring control mode (R31). Specify the same commands as that for master spindle.
2 The setting of grid shift amount in Cs contour control (parameter No. 4135 (Refer to “Parameters”)) is valid.

(6) Constant Surface Speed Control
Constant surface speed control keeps the surface speed constant at the contact point of the tool and the work-piece.

- Start of Constant Surface Speed Control L_ S_ IP=_ R41;
The spindle goes under the constant surface speed control at the reference time (L_). Surface speed is specified in S address. The unit is m/min or feet/min. The standard axis and position is specified IP=_.
If bit 0 (PCA) of parameter No. 11104 is 0, the standard position (IP=_) is the position of machine coordinate system. The tool offset value is not considered for the calculation of constant surface speed.
If bit 0 (PCA) of parameter No. 11104 is 1, the standard position (IP=_) is the position of workpiece coordinate system. Standard position is commanded as 0 usually because the tool offset value is considered for the calculation of constant surface speed.

The standard axis name can be specified with not only one address word such as X, but also one address word and path(head) number such as X2. The servo axis in other path(head)s can be commanded as the standard axis even if the expanded axis name (parameter No.1025, No.1026) is not applied.
The spindle rotates in the direction of CCW (counterclockwise) when minus value is specified as surface speed.

Example)
L100 S4000 X2=100.0 R41 ;
The spindle control is changed into the constant surface speed control mode.
The standard axis of the constant surface speed control is X2.
The standard position is 100.0 (radius designation).The spindle is switched to the constant surface speed control mode.

The standard axis of constant surface speed control can be specified as follows:
- L_ S(surface speed) [one character of axis name in a certain path]=[machine coordinate value (radius value) corresponding to workpiece coordinate 0] R41 ;
When the length of axis name is 1, the axis is recognized as the axis in the same path as the spindle.
- L_ S(surface speed) [standard axis + number of the path]=[machine coordinate value (radius value) corresponding to workpiece coordinate 0] R41 ;
The axis in the other path can be specified as the standard axis without using the extended axis name function (parameter No. 1025, 1026).

By the above function, it is not necessary to use the extended axis name function in Path Table Operation. Use the extended spindle name for the spindle name.
Example)
It is assumed that the axis name in each path is set as follows.
Path 1: X, Z, C, Y Spindle name S1, S11
Path 2: X, Z, C, Y Spindle name S2, S21
Path 3: XC, ZC, C, YC Spindle name S3, S31

<TIME_TABLE_1001_S1> ; Spindle command table of 1st spindle in path 1

```
R98 ;
L1000 S200    X2=50.0 R41 ;    The standard axis is X in path 2
                               The standard coordinate 50.0 (100.0 as diameter value)
                               The surface speed 200 m/min
L4000 R40 ;
L5000 S-200 X=50.0 R41 ;       The standard axis is X in path1
                               The surface speed -200 m/min
L8000 R40 ;
L9000 S200    XC=50.0 R41 ;    The standard axis is XC in path3 (extended axis name)
                               The surface speed 200 m/min
L12000 R40 ;
R99 ;
```

NOTE
1 Constant surface speed signal CSS does not become ”1” during the constant surface speed control in Path Table Operation.
2 Specify the standard coordinate (IP=_) as same of the number of decimal part digits for least input inclement (3-digits in the decimal part with IS-B, 4-digits with IS-C). Conversion alarm occurs if the specified value exceeds the number of digits.

- Clamp Speed of Constant Surface Speed Control L_ S_ R42 ;
  The clamp speed of the constant surface speed control is specified with S address. The unit is min^{-1}. The actual spindle speed in the constant surface speed control (R41) is limited by this clamp speed. The speed is effective till the CNC power turns off.
  When bit 5 (SOC) of parameter No. 3708 is 0, the spindle override is carried out after clamping the speed.
  And, when bit 5 (SOC) of parameter No. 3708 is 1, the spindle override is carried out before clamping the speed.
  It does not affect the constant surface speed control by G96.

- End of Constant Surface Speed Control L_ R40 ;
  The constant surface speed control is cancelled at the reference time (L_), and returns to the speed control mode. The spindle speed returns before starting of constant surface speed control.

- Required time interval for switching each mode

To switch the spindle control mode, free time is required. The required time interval depends on the current spindle control mode and the specified spindle command. Required time interval is shown in the following table. Specify the next command after the intervals as shown in the following table.

Table 20.7 (d) Required time interval for switching each mode

Current spindle control mode	Effective command for spindle	Required time interval to complete the command / Movement when the command is specified
Spindle Speed Control mode (R11)	R11 Spindle speed control	4msec / Spindle speed is renewed, and acceleration or deceleration control to specified speed is started.
	R21 Start of spindle synchronization	Until following up to the speed of the master spindle / The mode is switched to spindle synchronization mode, and acceleration or deceleration to follow up to the speed of the master spindle is started.
	R31 Start of contouring control	Until the spindle completes positioning to the specified position / Positioning to the specified position is started, and after positioning finished, the mode is switched to contouring control mode.
	R41 Start of constant surface speed control	4msec / The mode is switched to constant surface speed control mode, and acceleration or deceleration to the constant surface speed is started.
	R42 Clamp speed of constant surface speed control	4msec / Clamp speed of constant surface speed control is renewed.
Spindle Synchronization mode (R21)	R11 Spindle speed control	4msec / Spindle speed at the end of spindle synchronization is set.
	R20 end of spindle synchronization	4msec / The mode is switched to spindle speed control, and acceleration or deceleration to the spindle speed before start of spindle synchronization is started.
	R42 Clamp speed of constant surface speed control	4msec / Clamp speed of constant surface speed control is renewed.
Contouring Control mode (R31)	R11 Spindle speed control	When the spindle is stopped: 4msec When the spindle speed is decelerating by acceleration/deceleration after interpolation for previous contouring control command: Time constant (parameter No.11110 (refer to “Parameters”)) + 8msec When the next command is R31(start of contouring control): 8msec / The mode is switched to spindle speed control, and acceleration to the specified speed is started.
	R21 Start of spindle synchronization	Time for following to the speed of the master spindle / The mode is switched to Spindle Synchronization, and acceleration & deceleration control to following to the speed of the master spindle is started.
	R30 End of contouring control	When the spindle is stopped: 4msec When the spindle speed is decelerating by acceleration/deceleration after interpolation for previous contouring control command : Time constant (parameter No.11110 (refer to “18.Parameters”)) + 8msec / The mode is switched to spindle speed control, and deceleration is started to stop the spindle.
	R42 Clamp speed of constant surface speed control	4msec / Clamp speed of constant surface speed control is renewed.
Constant Surface Speed Control mode (R41)	R11 Spindle speed control	4msec / Surface speed is renewed.
	R41 Start of constant surface speed control	4msec / Surface speed is renewed.
	R42 Clamp speed of constant surface speed control	4msec / The mode is switched to spindle speed control.
	R40 End of constant surface speed control	4msec / Switching occurs to the speed control mode.

NOTE
Required time interval until the commands are completed might be different depending on the series of CNC system software.

Example 1)

The following example is when R42 command is specified during spindle speed control (R11) mode. In this case, it takes 4msec to complete R42 command as shown in the above table. Therefore, specify next R41 command at 4msec after the previous R42 command is specified.

(Spindle speed control(R11) mode) The mode of the spindle before command specified
L_ S_ R42(Clamp speed) Spindle command
L_ S_ IP=_ R41(Start of constant surface speed control)

Example 2)

The following example is when R11 command is specified during Cs contour control (R31) mode. In this case, It takes 4msec or time constant (parameter No. 11110) + 8msec to complete R11 command as shown in the above table. Therefore, specify next R11 command at 4msec or time constant (parameter No. 11110) + 8msec after the previous R11 command is specified.

(Contouring control mode (R31)) The mode of the spindle before command specified
L_ S_ R11 (Speed control mode) Spindle command
L_ S_ (R11) (Speed control mode)

Data of Auxiliary Function Table

The auxiliary function (M code) can be output during the Path Table Operation by using the Auxiliary Function Table. The reference value and the auxiliary function are specified in order as well as the data of axis motion table. Three M codes can be output at once.
The auxiliary function table can be commanded for each path(head) respectively. For example, one auxiliary function table can be executed in each path(head) at three path(head) system.
B code output is not available in the auxiliary function table.

- Format

L_ Maa (Mbb) (Mcc) ;

L : Reference Value
Maa : First M code
Mbb : Second M code
Mcc : Third M code
The second M code and third M code can be omitted.

Example)

When the reference value is L1000, M10 is output. When it is L2000, M11 and M12 are output. And when it is L3000, M20, M21 and M22 are output simultaneously.

```
<TIME_TABLE_0001_M>;
R98 ;
:
L1000 M10 ;
L2000 M11 M12 ;
L3000 M20 M21 M22 ;
:
```

(1) Reference Value

The reference value is designated in L address. The value should increase gradually. The range of designation is as follows.
Time Reference, the axis position reference and the spindle position reference should be commanded with the absolute value.

Valid data range

Time Reference:L0 to L99999999.999999

(a unit is msec, it is specified in the multiple of 1. A decimal point can be specified, but it is rounded off.)

Axis position Reference:L-999999999 to L999999999

(Unit is least command increment.)

If bit 0 (PCA) of parameter No. 11104 = 0, coordinate of machine coordinate system is commanded.

If bit 0 (PCA) of parameter No. 11104 = 1, coordinate of workpiece coordinate system is commanded.

Spindle position Reference:L-999999999999.999 to 999999999999.999 degree

The reference value should increase gradually.

NOTE

1 For details of the second M code and third M code, refer to "MULTIPLE M COMMANDS IN A SINGLE BLOCK" in this manual.

2 When M code is output and the ladder of the PMC does not execute the completion processing of the previous M code, alarm PS0451 “ILLEGAL AUXILIARY FUNCTION TABLE COMMAND” is issued.

3 When the time reference override is 100%, M Codes in auxiliary function table are output in each 4msec or 8msec (depends on the series of the system software).
Therefore, when the following commands are specified, all M37 command is output to PMC in the same time.

```
L1.0 M37
L2.0 M37
L3.0 M37
L4.0 M37
```

However, only L0 is treated exceptionally, and means "Start of Path Table Operation".

Table Footer

At the end of the Path Table, “R99” should be specified. If the Path Table number to be connected in Q address is specified on the header of the Path Table, the specified Path Table is executed continuously.

- Format

```
R99 ;
```

R99 : Definition of Table Footer

Convert Path Table to Executive form data

Before the Path Table Operation, the Path Tables stored in the part program files should be converted to the executive form data.

(1) The conversion is triggered by asserting Path Table Request signal PTREQ, or by commanding through C Language Executor or FOCAS2 window library function cnc_startptcnv().

(2) When the request signal is accepted, CNC sets Path Table Conversion Executing signal PTCNV to “1”, and starts conversion. CNC sets Path Table Ready signal PTRDY to “0” and Path Table Conversion Alarm signal PTCVA to “0”.

(3) After all the Path Tables in the path(head) are converted, CNC sets PTCNV to “0”, and Path Table Ready signal PTRDY to “1”. When alarms are detected during the conversion, PTCNV is set to “0”, and Path Table Conversion Alarm signal PTCVA is set to “1”.

NOTE
If conversion from Path Table to executive form data ends abnormally, conversion alarm number can be gotten by the C Language Executor or FOCAS2 window library function cnc_rdptcnvalm(). For details, refer to "CNC/PMC window library" of "C Language Executor Programming Manual" (B-63943EN-3).

A conversion request can be received from all path(head)s at the same time. However, actual conversion is performed in order of each path(head). While conversion of one path(head) is executed, other path(head)s conversion is kept waiting.
The conversion time changes to depend on various conditions (e.g. part program, number of path(head), number of axis, operation state, CPU load and etc.).
It is prohibited to download the Path Table programs during converting the Path Tables.

- Conversion Sequence

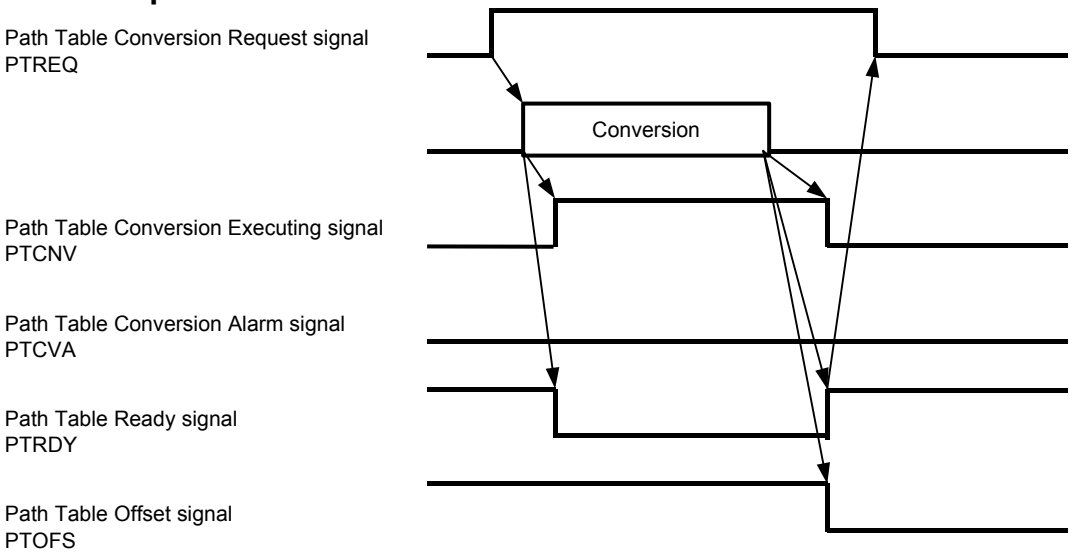


Fig. 20.7 (f) Sequence when conversion ends normally

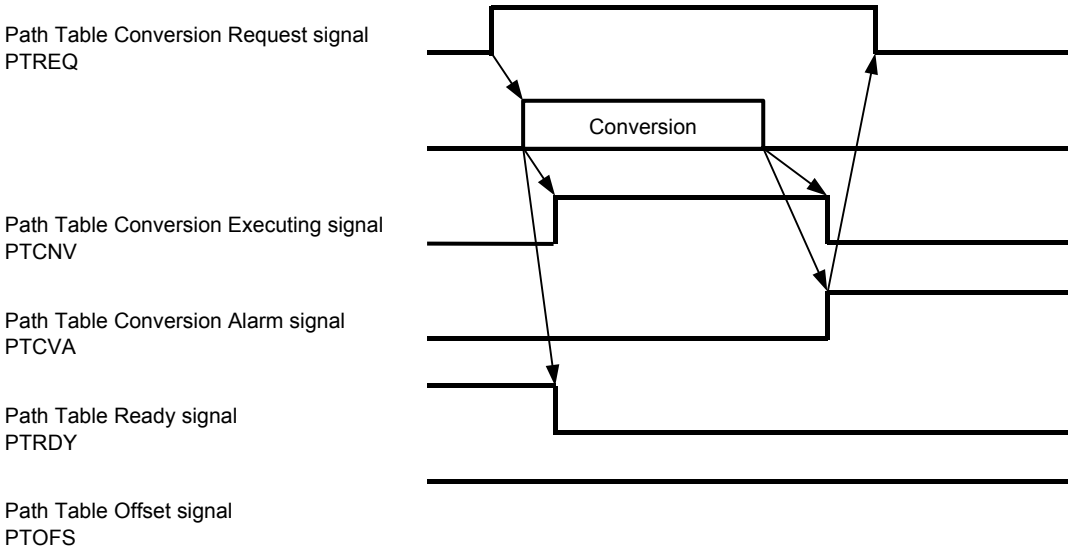


Fig. 20.7 (g) Sequence when conversion ends abnormally

Execution of Path Table

The M code, which is specified in parameter No.11100, starts the Path Table Operation. During the Path Table Operation, Path Table Mode signal PTMOD is turned "1".

- Format

M_ (P_) Q_ ;

M_	:	M code to start the Path Table Operation
P_	:	Wait condition
Q_	:	Path Table number

Example 1)

Mxx Q0001 ;

Path Table No.1 is started without path(head) waiting.

Example 2)

Mxx P123 Q0001 ;

After waiting path(head) 1, 2, and 3, Path Table No. 1 is started synchronously.

Because there are no restrictions on the order in which the numeric characters are specified, the following six possible values can be specified:

P123, P132, P213, P231, P312, P321

Path numbers specified in combination in different orders for different path(head)s are effective as long as the numbers of the relevant path(head)s are specified.

Example)

P123 for path(head) 1, P231 for path(head) 2, and P321 for path(head) 3.

These are treated as the same P value and these path(head)s can be made to wait for one another.

To make path(head) 10 and another path(head) wait for each other, specify a value of 0 for the combination.

If a number begins with 0, 0 cannot be recognized. Specify 0 in the second or subsequent digit from the left.

Incorrect Example) P013579

Correct Example) P103579

In the incorrect example, the P value is assumed to be the same as P13579. Path 10 cannot be made to wait for other path(head)s and other path(head)s cannot be made to wait for path(head) 10.

NOTE

1 Other commands should not be commanded together with M code to start the Path Table Operation.
2 M code to start the Path Table Operation is not output to PMC.
3 Setting value of parameter No. 11100 should be out of range of the minimum value (parameter No. 8110) and the maximum value (parameter No. 8111) of waiting M code.

Transit between ISO Program and Path Table Operation

- ISO Program to Path Table Operation

  The constant surface speed control mode is cancelled.

  The spindle speed and spindle rotation direction is preserved.

  The spindle synchronization is maintained.

  If the spindle is not in SV speed control mode, spindle becomes contour control mode (R31).

- Path Table Operation to ISO Program
  The constant surface speed control mode is cancelled.
  The spindle speed is preserved.
  To preserve the spindle rotation direction, adjust the rotation direction with SV reverse signal SVRVS1 to SVRVS8.
  To maintain the spindle synchronization mode, set the spindle synchronization control signal SPSYCs1 to SPSYCs4 to "1" by the condition of SSYO1 to SSYO4.
  If the spindle synchronization control signal SPSYCs1 to SPSYCs4 is set to "0", the spindle synchronization mode is cancelled after the Path Table Operation ends.
  Contour control mode is maintained.

The Path Table numbers at start should be the same for all axes in the same path(head). The Path Table number to be connected can be specified with different number for each axis.
After the entire connected Path Tables are finished, the CNC block next to the Path Table start M code is executed. It takes several dozen msec for shifting from the conventional CNC program to the Path Table Operation.

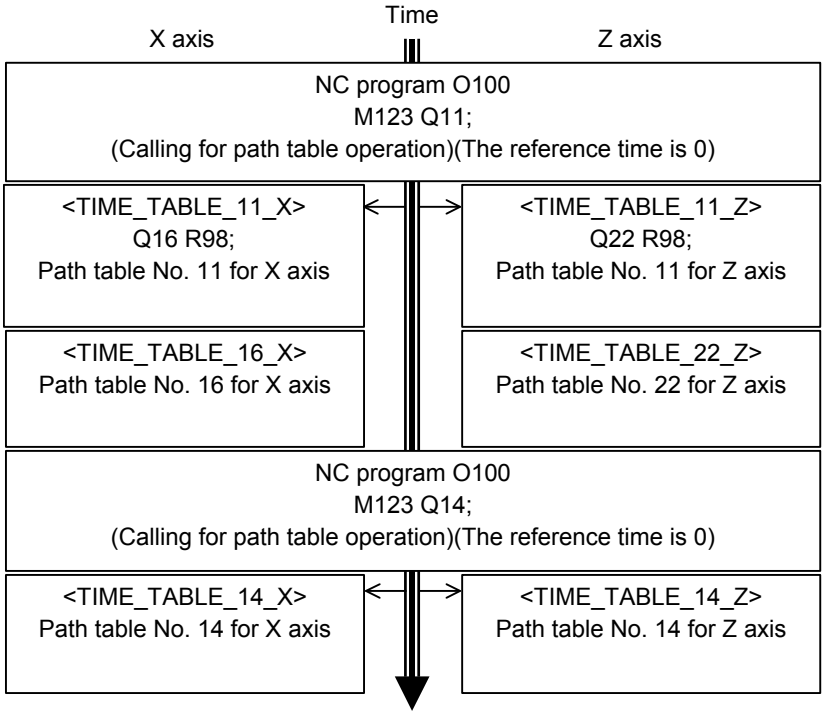


Fig. 20.7 (h) Simultaneous control of multiple axes in Path Table Operation

NOTE
The axis coordinate at the start of Path Table operation and the axis command on the Path Table is checked. If the difference exceeds the parameter No.11101, alarm PS0442 “PATH TABLE COMMAND EXCES ERR” is issued.

Path Table Operation in Multi-path(head)s

The independent Path Table Operation is available in each path(head). However, the reference time is common. Therefore, in case of the time reference, the Path Table Operations with synchronizing among multi-path(head)s are possible.
The reference time is not reset at the end of each Path Table Operation. The reference time is reset after all of Path Table Operations are finished.

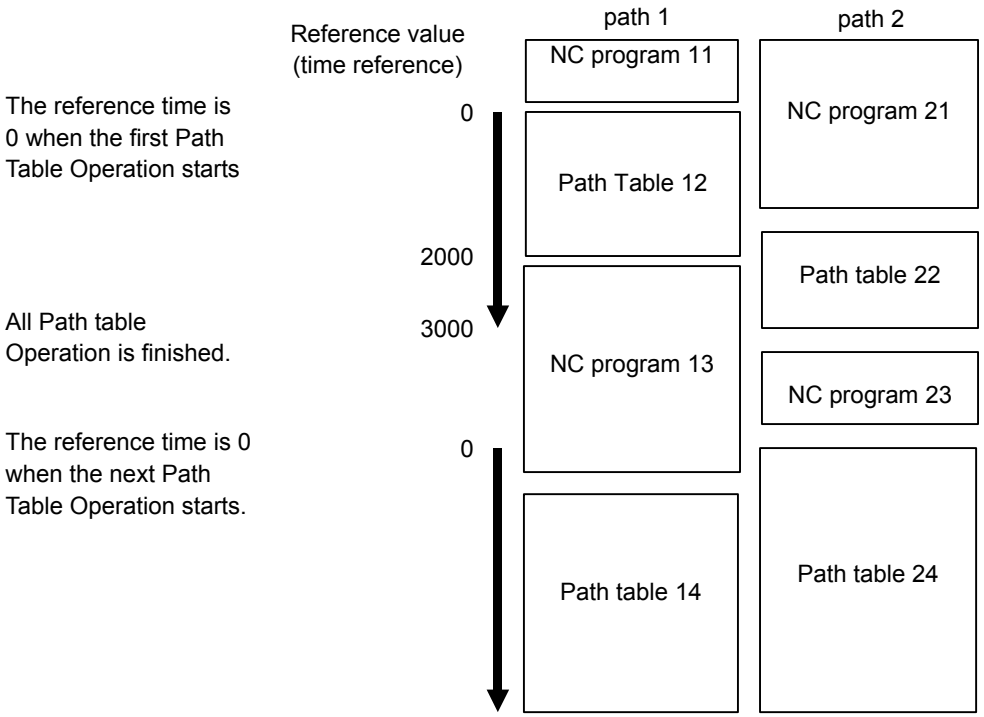


Fig. 20.7 (i) Path Table Operation in multi-path(head)s

NOTE
It is necessary when an alarm occurred a certain path(head) during the Path Table Operation and when it is reset to reset the path(head) all that a Path Table Operation was carried out. So, Set bits 0 (RST) and 1 (IAL) of parameter No.8100 to 0.

Override

- Time reference override
  Override is available for the time reference by asserting Path Table Override signals PTOV1 to PTOV7.
  The range of the override is from 0% to 100% with 1% increment.
- Spindle override
  Spindle override is effective during Path Table Operation.
- Feedrate override
  Feedrate override is not available during Path Table Operation.
- Rapid traverse override
  Rapid traverse override is not available during Path Table Operation.

Example

It is assumed that the following axis configuration and the following parameter are set.

- Least input increment: IS-B (metric input)
- Servo axes: X (diameter designation), Z (radius designation)
  Cs contour controlled spindle:C (spindle S)
- M code for Path Table Operation (parameter No.11100) =1111

Behavior when the following program O0001 is executed on this system is as follows:

NC program

```
O0001 ;
N1 G00 X100.0 Z0.0 ;
N2 M1111 Q0001 ;
N3 M30 ;
```

Axis motion table for the X axis

```
<TIME_TABLE_0001_X> ;
Q0002 R98 ;
L0 X50.0 ;
L1000 X75.0 ;
L2000 X50.0 ;
R99
```

```
<TIME_TABLE_0002_X> ;
K1 P0003 R98 ;
R99
```

```
<TIME_TABLE_0003_X> ;
R98 ;
L3000 X0.0
R99
```

Axis motion table for the Z axis

```
<TIME_TABLE_0001_Z> ;
Q0002 R98 ;
L0 X0.0 ;
L1000 X50.0 ;
R99
```

```
<TIME_TABLE_0002_Z> ;
R98 ;
L2000 X150.0 ;
L3000 X0.0 ;
R99
```

Spindle command table for the spindle S

```
<TIME_TABLE_0001_S> ;
Q0002 R98 ;
L0 S600 ;
L1000 X0.0 R31 ;
L2000 X180.0 ;
R99
```

```
<TIME_TABLE_0002_S> ;
R98 ;
L3000 X720.0 ;
R99
```

Auxiliary function table

```
<TIME_TABLE_0001_M> ;
R98 ;
L0 M100 ;
L1000 M200 ;
L2000 M300 ;
R99
```

```
<TIME_TABLE_0002_M> ;
R98 ;
L3000 M400 ;
R99
```

(1) N1 block is executed, and axes are positioned at X=100.000 mm and Z=0.000 mm.
(2) N2 block is executed, and Path Table No. 1 is called.
(3) Table 20.7 (e) lists the behavior of each axis in Path Table Operation. (Because the X axis is diameter designation, its travel distance is doubled.)

Table 20.7 (e) Behavior in Path Table Operation (Time reference override is 100%)

Reference value (time reference)	Behavior of the X axis	Behavior of the Z axis	Behavior of the spindle S	M code
0	Current pos.: 100.0 mm Aim to reach 150.0 mm 1000 msec later (6,000 mm/min)	Current pos.: 0.0mm Aim to reach 50.0 mm 1000 msec later (6,000mm/min)	Starts rotating at 600 min^{-1} (Clockwise)	M100 is output.
1000	Current pos.: 150.0mm Aim to reach 100.0 mm 1000 msec later (-6,000mm/min)	Current pos.: 50.0mm Connects to table 2. Aim to reach 150.0 mm 1000 msec later (12,000mm/min)	Starts moving to 0.0 degrees. Switched to contour control mode. Aim to reach 180.0 deg. 1000msec later	M200 is output.

Reference value (time reference)	Behavior of the X axis	Behavior of the Z axis	Behavior of the spindle S	M code
2000	Current pos.: 100.0mm Connect to Path Table No. 2. Branch on a value of PTSK1 (Gn522.0). - If PTSK = "0": Path table operation ends (no Path Table Number to be connected specified). - If PTSK = "1": Connect to Path Table No. 3. Aim to reach 0.0 mm 1000 msec later (-12,000mm/min)	Current pos.: 150.0mm Aim to reach 0.0 mm 1000 msec later (-18,000mm/min)	Current pos.: 180.0deg. Connect to Path Table No. 2. Aim to reach 720.0 degrees 1000 msec later ($90min^{-1}$)	M300 is output. Path Table Operation ends (no Path Table Number to be connected specified).
3000	Only when connected to table No. 3 Current pos.: 0.0mm Path Table Operation ends (no Path Table Number to be connected specified).	Current pos.: 0.0mm Path Table Operation ends (no Path Table Number to be connected specified).	Current pos.: 720.0 deg. (=0.0 deg.) Path table operation ends (no Path Table Number to be connected specified).	

(4) NC program ends because of M30 in the N3 block.

Limitation

(1) Servo Guide
Servo condition such as a torque command, actual speed, actual position during the Path Table Operation can be watched by the Servo Guide or the Servo Guide Mate.

(2) Spindle speed change time
The change time when a spindle speed changes can be calculated based on the theory value and the actual measurement value.

(3) Tool offset
The amount of tool offset at the start of Path Table Operation is held until the Path Table Operation completion

(4) Threading
The threading can be achieved by using spindle position reference. The multi-threading with one spindle can be executed in Path Table Operation. The multi-threading with different spindles can be executed at the same time in Path Table Operation.

(5) Excess error (Servo alarm)
The Path Table data should be specified with considering the position deviation value.

(6) Axis speed control
The axis speed is not automatically limited in Path Table Operation. The axis speed is output according to commanded time and axis position in Path Table.

(7) Stop operation
The stop operation is carried out by decreasing override to zero gradually and reset operation.

(8) Return to reference point
"Return to reference point" needs to be commanded in ISO code.

(9) Polygon machining with two spindles
Polygon turning with two spindles is not available in Path Table Operation.

(10) Languages
The following 15 languages are supported.
English, Japanese, German, French, Spanish, Italian, Chinese, Korean, Portuguese, Dutch, Danish, Swedish, Hungarians, the Czech and Polish

(11) Servo off
On the following conditions, servo off can be used in Path Table Operation.
-Spindle
- Follow-up is not performed.
- Speed commands are not specified in spindle speed control (R11) mode (Spindle Stop or S0)
- Movement commands are not specified in Cs contour control (R31) mode.

-Servo axis
- Follow-up is not performed.
- Movement commands are not specified.

(12) Rotary axis
It doesn't support the shortcut of roll over and rotary axis control. A rotation direction of the rotation axis in the Path Table follows the sign of the command value.

Example)
In the case of the rotation axis (type A) that bit 0 (ROTx) of parameter No. 1006 is set to 1 and bit 1 (ROSx) of parameter No. 1006 is set to 0, when the following program is execute, the rotation axis is moved as follows.

```
O3000 ;
G96.1 P11 R0 ;
G0 X0 Z0 C0 ;
M100 Q5555 ;
G04 X1. ;
M30 ;

<TIME_TABLE_5555_S1>
R98
L0 X0 R31 ;        - -> 0 [degree]
L10000 X500. ;   - ->  360 [degree]+140 [degree] = 500 [degree] move
L20000 X-500. ; - ->  -360 [degree]+-360 [degree]+-280[degree] = -1000 [degree] move
L30000X0 ;        - ->  360 [degree]+140 [degree] = 500 [degree] move
R99
```

(13) Manual absolute
When manual intervention is done during the Path Table Operation block, an axis is moved in the position where it shifted until the Path Table Operation is finished.
Even when a manual absolute is effective, an axis is moved in the position where the one for the manual intervention quantity shifted during the Path Table Operation. The amount of manual intervention is pulled in the absolute command block after the Path Table Operation completion.
Example)

```
N10 G00 X0.0 Z0.0 ;
N20 G00 X2.0 Z2.0 ;
N30 M100 Q1111 ;
N40 G00 X2.0 Z5.0 ;

<TIME_TABLE_1111_X>
R98 ;
L0 X1.0 ;        (The case that X axis is radius designation, X2.0)
L1000 X1.5 ;   (The case that X axis is radius designation, X3.0)
L2000 X2.0 ;   (The case that X axis is radius designation, X4.0)
R99 ;
```

```
<TIME_TABLE_1111_Z>
R98 ;
L0 X2.0 ;
L1000 X3.0 ;
L2000 X4.0 ;
R99 ;
```

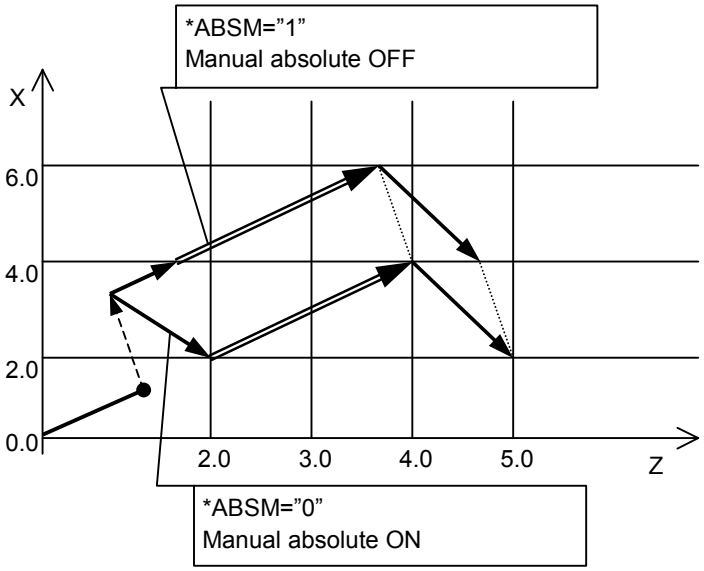


Fig. 20.7 (j) Manual intervention performed en route to CNC block in front of Path Table Operation

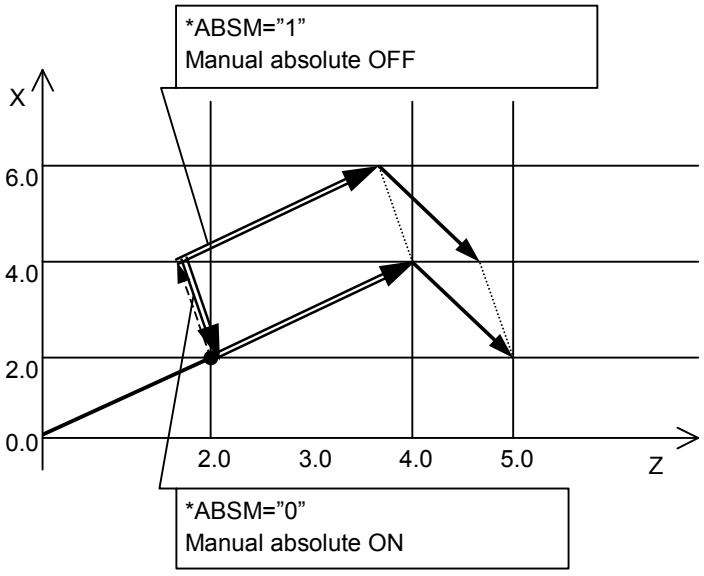


Fig. 20.7 (k) When it started from the path table operation after the manual intervention

NOTE

If Path Table Operation is started when manual absolute is effective, the axis which is move with manual intervention might move rapidly. For the safety ,set allowable value on parameter No.11101. When the amount of manual intervention is bigger than allowable value, alarm PS0442, ”PATH TABLE COMMAND EXCES ERR” is issued.

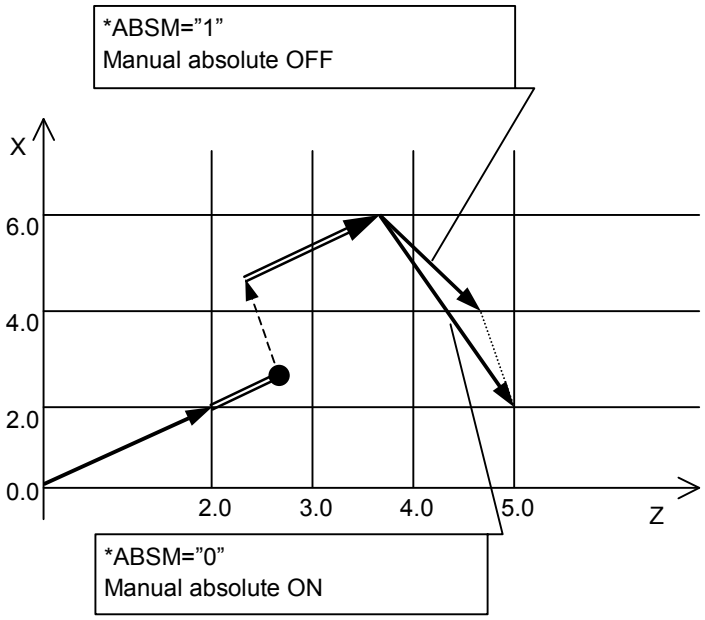


Fig. 20.7 (l) When a manual intervention performed en route to Path Table Operation

(14) Machine lock
If bit 0 (PCA) of parameter No. 11104 is 1, when machine lock signal or each axis machine lock signal is set to "1", the Path Table Operation can execute.
If bit 0 (PCA) of parameter No. 11104 is 0, when machine lock signal or each axis machine lock signal is set to "1", the Path Table Operation of all path(head)s stop.

(15) Interlock
If an axis which moves by Path Table Operation is in the state of the interlock, Path Table Operation of all path(head)s stop.

(16) Inch/mm
When an input unit in the Path Table Operation start is different from the input unit when the Path Table converted, alarm PS0452, “ILLEGAL PATH TABLE OPERATION” (detail alarm No. 33) is issued.

(17) Mirror image
If bit 0 (PCA) of parameter No. 11104 is 1, a mirror image can be applied by the setting screen or the mirror image signal MI1 to MI8 during the Path Table Operation.
If bit 0 (PCA) of parameter No. 11104 is 0, mirror image during the Path Table Operation becomes invalid.

(18) Tool geometry/wear compensation
The tool geometry/wear compensation which compensated in NC block before the Path Table Operation is effective during the Path Table Operation.

(19) Tool length compensation (G43)
Tool length compensation (G43) which compensated in NC block before the Path Table Operation is effective during the Path Table Operation.

(20) Tool nose radius compensation, cutter compensation C
The Path Table Operation can't be started during the tool nose radius compensation mode and the cutter compensation C mode. Alarm PS0452 (detail alarm No. 34) is issued.

(21) Programmable mirror image (G51.1)
The Path Table Operation can't be started during the programmable mirror image (G51.1). Alarm PS0452 (detail alarm No. 34) is issued.

(22) Mirror image for double turret (T:G68)
The Path Table Operation can't be started during the mirror image for double turret (T:G68). Alarm PS0452 (detail alarm No. 34) is issued.

(23) Coordinate system rotation (M:G68, T:G68.1), 3-dimensional coordinate system conversion (M:G68, T:G68.1)
The Path Table Operation can't be started during the coordinate system rotation (M:G68,T:G68.1) or 3-dimensional coordinate conversion(M:G68,T:G68.1). Alarm PS0452 (detail alarm No. 34) is issued.

(24) Scaling
The Path Table Operation can't be started during Scaling. Alarm PS0452 (detail alarm No. 34) is issued.

(25) Arbitrary angular axis control
If bit 0 (PCA) of parameter No. 11104 is 1, Path Table Operation can be performed for angular axes.
If bit 0 (PCA) of parameter No. 11104 is 0, Path Table Operation cannot be performed for angular axes. Alarm PS0452 (detail alarm No. 35) is issued.

(26) Tool axis direction control (G53.1)
The Path Table Operation can't be started during the tool axis direction control (G53.1). Alarm PS0452 (detail alarm No. 34) is issued.

(27) Tool center point control (G43.4, G43.5)
The Path Table Operation can't be started during the tool center point control (G43.4,G43.5). Alarm PS0452 (detail alarm No. 34) is issued.

(28) Tool length compensation in tool axis direction (G43.1)
The Path Table Operation can't be started during the tool length compensation in tool axis direction (G43.1). Alarm PS0452 (detail alarm No. 34) is issued.

(29) Polar coordinate command (G16)
The Path Table Operation can't be started during the Polar coordinate command (G16). Alarm PS0452 (detail alarm No. 34) is issued.

(30) Parameter setting in Path Table Operation
In Path Table Operation, set parameters with PMC window function which is triggered by M code in the auxiliary function table. Transition of servo parameters is executed by PMC window function even in automatic operation if all axes are stop. If any axes are moving, PMC window function returns the error No.113 as completion code.

Note

(1) This function is an optional function.

(2) In order to control spindle in Path Table Operation, following optional functions are necessary.
- Spindle control with Cs contour control
- Cs contour control

(3) When two or more than spindle are used, following optional function is necessary. And also, the following optional function is required when the master spindle of the synchronous control or the spindle position reference is commanded with the extended spindle name in the Path Table Operation.
- Multi spindle control

(4) The following optional functions are necessary to use spindle synchronous control and constant surface speed control in Path Table Operation.
- Spindle synchronous control
- Constant surface speed control

21 AXIS CONTROL FUNCTIONS

Chapter 21, "AXIS CONTROL FUNCTIONS", consists of the following sections:

21.1 AXIS SYNCHRONOUS CONTROL....607
21.2 POLYGON TURNING (G50.2, G51.2)....609
21.3 SYNCHRONOUS, COMPOSITE AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND (G50.4, G51.4, G50.5, G51.5, G50.6, AND G51.6)....614
21.4 ROTARY AXIS ROLL-OVER....617
21.5 TOOL RETRACT AND RECOVER....619
21.6 ELECTRONIC GEAR BOX....626
21.7 TANDEM CONTROL....661
21.8 PIVOT AXIS CONTROL....661
21.9 CHOPPING FUNCTION....664
21.10 SKIP FUNCTION FOR FLEXIBLE SYNCHRONOUS CONTROL....671
21.11 CHOPPING FUNCTION BY FLEXIBLE SYNCHRONOUS CONTROL....673
21.12 HIGH PRECISION OSCILLATION FUNCTION....674

21.1 AXIS SYNCHRONOUS CONTROL

When a movement is made along one axis by using multiple servo motors as in the case of a large gantry machine, a command for one axis can drive the multiple motors by synchronizing one motor with the other. An axis used as the reference for axis synchronous control is called a master axis (M-axis), and an axis along which a movement is made in synchronism with the master axis is called a slave axis (S-axis).

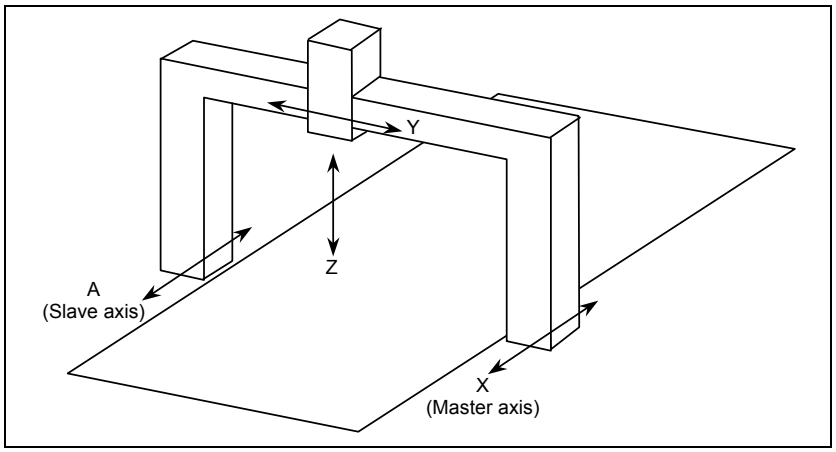


Fig. 21.1 (a) Example of machine with X and A being synchronous axes

Operation where axis synchronous control is turned on (enabled) to make a movement along the slave axis in synchronism with the master axis is called a synchronous operation. Operation where axis synchronous control is turned off (disabled) to move a master axis and a slave axis independently is called a normal operation.

(Example)

Automatic operation when the master axis is the X-axis and the slave axis is the A-axis:
In synchronous operation, both the X-axis and the A-axis are moved according to the programmed command Xxxxx for the master axis.

In normal operation, the master axis and the slave axis are moved independently as in the case of normal CNC control. The programmed command Xxxxx makes a movement along the X-axis. The programmed command Aaaaa makes a movement along the A-axis. The programmed command Xxxxx Aaaaa makes movements along the X-axis and A-axis at the same time.

For details, refer to the relevant manual published by the machine tool builder.

Note

NOTE

1 During axis synchronous control, a movement based on the reference position return check (G27), automatic reference position return (G28), 2nd/3rd/4th reference position return (G30), or machine coordinate system selection (G53) command is made as described below according to the setting of bit 2 (SRF) of parameter No. 8305.
   <1> When SRF = 0, the same movement as made along the master axis is made along the slave axis.
   <2> When SRF = 1, a movement is made along the slave axis to the specified position independently of a movement made along the master axis to the specified position.
2 A command not involving a movement along an axis such as the workpiece coordinate system setting command (G92) and local coordinate system setting command (G52) is set with the master axis according to the master axis programming.
3 When switching the synchronization state in a program, be sure to specify M codes (parameters Nos. 8337 and 8338) for turning synchronization on and off. By switching between the input signals SYNC <Gn138> and SYNCJ <Gn140> from the PMC with the M codes, the synchronization state can be switched in the program.
4 If a programmed command is specified for the slave axis during synchronous operation, an alarm PS0213 is issued.
5 A programmed command can be specified for the slave axis when switching between synchronous operation and normal operation is set to 0 (with bit 5 (SCAx) of parameter No. 8304 set to 0) and the signal SYNC/SYNCJ select normal operation.

21.2 POLYGON TURNING (G50.2, G51.2)

Polygon turning means machining a workpiece to a polygonal figure by rotating the workpiece and tool at a certain ratio.

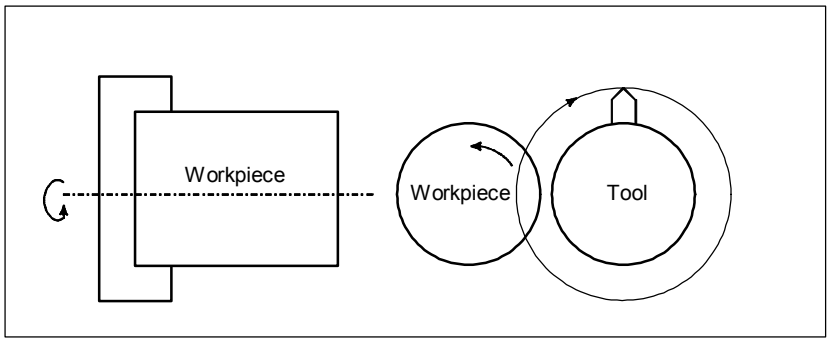


Fig. 21.2 (a) Polygon turning

By changing conditions which are rotation ratio of workpiece and tool and number of cutters, the workpiece can be machined to a square or hexagon. The machining time can be reduced as compared with polygonal figure machining using the polar coordinate interpolation. The machined figure, however, is not exactly polygonal. Generally, polygon turning is used for the heads of square and/or hexagon bolts or hexagon nuts.

As the tool rotary axis, one of the following can be used:

- CNC controlled axis (servo axis)
- Second spindle (Two serial spindles are connected.)

Polygonal machining performed using a servo axis as the tool rotary axis is referred to as polygon turning. Polygonal machining performed using the second spindle as the tool rotary axis is referred to as polygon turning with two spindles.

Function name	Workpiece axis	Tool rotary axis
Polygon turning	Spindle (Either an analog spindle or serial spindle is usable. However, a detector equivalent to a position coder is required.)	Servo axis
Polygon turning with two spindles	Spindle (Serial spindle)	Spindle (Serial spindle)

Explanation

A CNC controlled axis (servo axis) is assigned to the tool rotary axis.

This rotary axis of tool is called Y-axis in the following description. As the workpiece axis (spindle), either a serial spindle or analog spindle can be used.

The Y-axis is controlled by the G51.2 command, so that the ratio of the rotation speeds of the spindle (previously specified by S-command) and the tool becomes the specified ratio.

When simultaneous start is specified by G51.2, the one-rotation signal sent from the position codes set on the spindle is detected. After one-rotation signal detection, the Y-axis is controlled using the rotation ratio of the spindle and Y-axis specified by P and Q. So, a position coder needs to be attached to the spindle.

This control will be maintained until the polygon turning cancel command is executed (G50.2). Polygon turning is cancelled by any of the following in addition to the G50.2 command:

(1) Power off
(2) Emergency stop
(3) Servo alarm
(4) Reset (external reset signal ERS, reset/rewind signal RRW, and RESET key on the MDI panel)
(5) Occurrence of alarms PS0217 to PS0221, PS0314, and PS05018

NOTE

1 Before polygon turning, reference position return operation on the Y-axis needs to be specified to determine the rotation start position of the tool. This reference position return operation is performed by detecting a deceleration limit as in the case of manual reference position return operation. (By setting bit 7 (PLZ) of parameter No. 7600, reference position return operation can be performed without detecting a deceleration limit.)
2 The rotation direction on the Y-axis is determined by the sign of Q, and is not affected by the rotation direction of the position coder.
3 Among Y-axis position indications, the indication of a machine coordinate (MACHINE) changes within the amount of movement for 0 to 1 revolution as a movement is made on the Y-axis. The absolute coordinate and relative coordinate are not updated. So, when specifying an absolute-position command for the Y-axis after polygon turning mode cancellation, set a workpiece coordinate system after reference position return operation.
4 For the Y-axis engaged in polygon turning, jog feed and handle feed are disabled.
5 For the Y-axis not engaged in polygon turning, a move command can be specified as in the case of other controlled axes.
6 The Y-axis engaged in polygon turning is not counted in the number of simultaneously controlled axes.
7 One workpiece must be machined using a fixed spindle speed until the workpiece is finished.
8 G50.2 is the G code for suppressing buffering.

⚠ CAUTION

1 During polygon turning, threading cannot be performed.
2 For the Y-axis engaged in synchronous operation, the signals below are valid or invalid:
Signals valid for the Y-axis
- Machine lock
- Servo-off

Signals invalid for the Y-axis
- Feed hold
- Interlock
- Override
- Dry run
(At dry run time, however, the one-rotation signal is not awaited in a G51.2 block.)

Format

G50.2 Polygon turning cancel

G51.2 P_ Q_ ;

P,Q: Rotation ratio of spindle and Y-axis
Specify range:
P: Integer from 1 to 999
Q: Integer from -999 to -1 or from 1 to 999
When Q is a positive value, Y-axis makes positive rotation.
When Q is a negative value, Y-axis makes negative rotation.

NOTE
Specify G50.2 and G51.2 in a single block.

Example

```
G00 X100. 0 Z20.0 S1000.0 M03 ;   (Workpiece rotation speed 1000 (min-1))
G51.2 P1 Q2 ;                     (Tool rotation start (tool rotation speed 2000 (min-1))
G01 X80.0 F10.0 ;                 (X-axis infeed)
G04 X2.0 ;                        (Waiting 2 seconds)
G00 X100.0 ;                      (X-axis escape)
G50.2 ;                           (Tool rotation stop)
M05 S0 ;                          (Spindle stop )
```

- Principle of polygon turning

The principle of polygon turning is explained below. In the figure (Fig. 21.2 (b)) below the radius of tool and workpiece are A and B, and the angular speeds of tool and workpiece are α and β. The origin of XY Cartesian coordinates is assumed to be the center of the workpiece.
Simplifying the explanation, consider that the tool center exists at the position P_o (A, 0) on the workpiece periphery, and the tool nose starts from position P_{to} (A-B, 0).

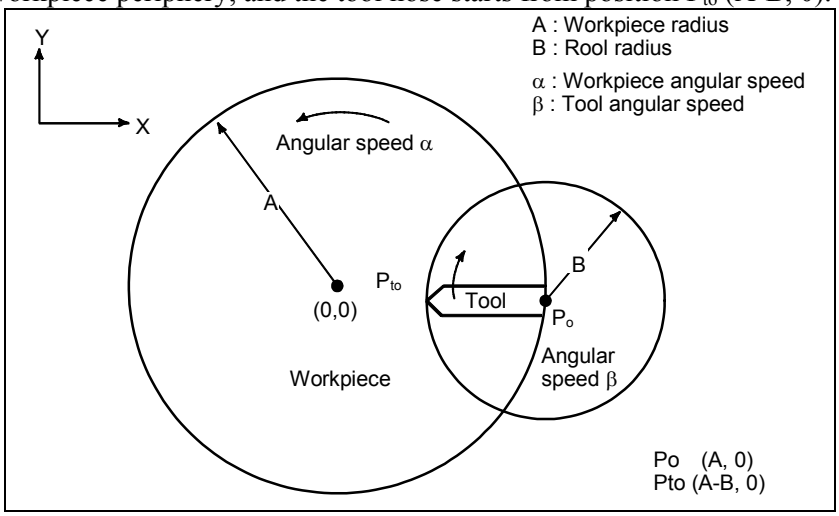


Fig. 21.2 (b) Principle of polygon turning

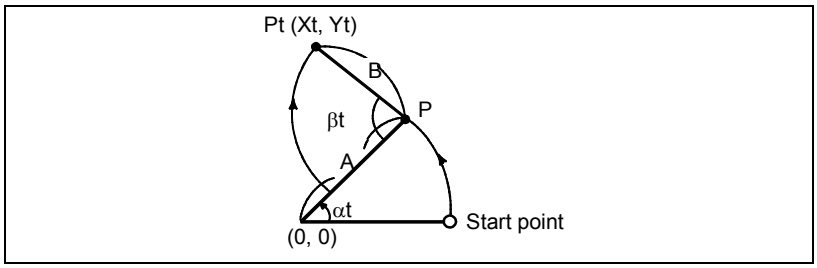


Fig. 21.2 (c) Tool nose position

In this case, the tool nose position Pt (Xt, Yt) after time t is expressed by equations 1 and 2:

$Xt=A\cos\alpha t-B\cos(\beta-\alpha)t$ (Equation 1)

$Yt=A\sin\alpha t+B\sin(\beta-\alpha)t$ (Equation 2)

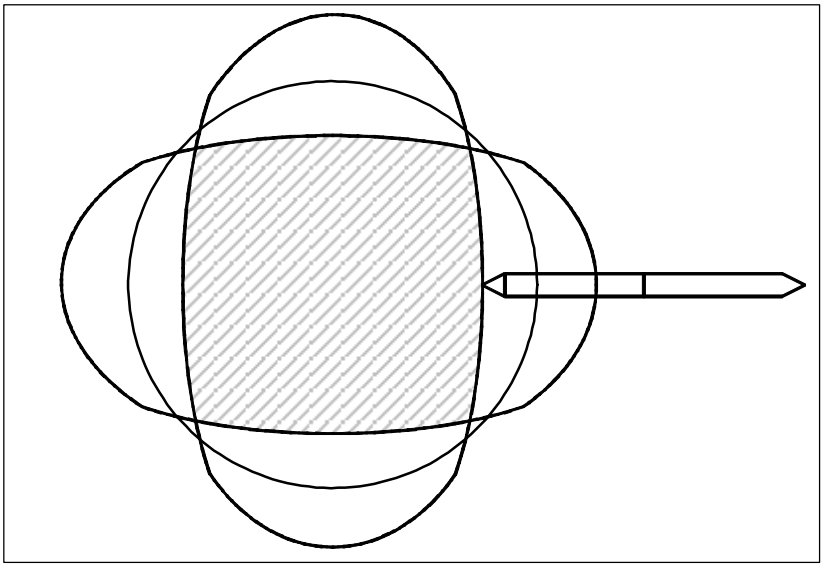
Assuming that the rotation ration of workpiece to tool is 1:2, namely, $\beta=2\alpha$, equations 1 and 2 are modified as follows:

$Xt=A\cos\alpha t-B\cos\alpha t=(A-B)\cos\alpha t$ (Equation 1)'

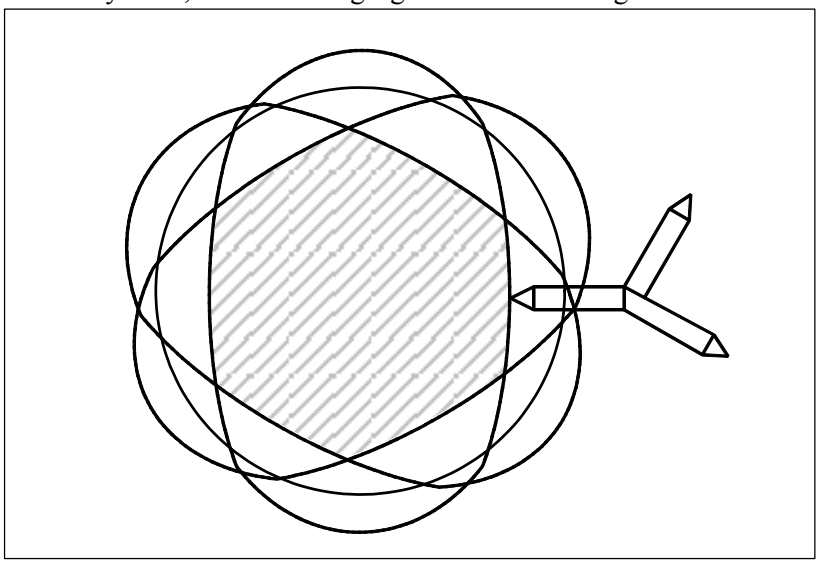
$Xt=A\sin\alpha t+B\sin\alpha t=(A+B)\sin\alpha t$ (Equation 2)'

These equations indicate that the tool nose path draws an ellipse with longer diameter A+B and shorter diameter A-B.

Then consider the case when one tool is set at 180° symmetrical positions, for a total of two. A square can be machined with these tools as shown below.



If three tools are set at every 120°, the machining figure will be a hexagon as shown below.



⚠ WARNING

For the maximum rotation speed of the tool, see the instruction manual supplied with the machine. Do not specify a spindle speed higher than the maximum tool speed or a ratio to the spindle speed that results in a speed higher than the maximum tool speed.

21.3 SYNCHRONOUS, COMPOSITE AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND (G50.4, G51.4, G50.5, G51.5, G50.6, AND G51.6)

Synchronous control, composite control, and superimposed control can be started or canceled using a program command instead of a DI signal. Synchronous control, composite control, and superimposed control based on a DI signal is also possible.

For the basic operations of synchronous control, composite control, and superimposed control, see Sections, "SYNCHRONOUS CONTROL AND COMPOSITE CONTROL" and "SUPERIMPOSED CONTROL" in the CONNECTION MANUAL (FUNCTION) (B-63943EN-1).

Format

G51.4 P_ Q_ (L_) ; Synchronous control start (L is omissible.)
G50.4 Q_ ; Synchronous control cancel

P: Synchronous master axis ID number
Q: Synchronous slave axis ID number

L: Parking command
1: Master parking (slave parking cancel)
2: Slave parking (master parking cancel)
0: No parking (parking cancel)
(When L is omitted, the specification of L0 is assumed.)

G51.5 P_ Q_ ; Composite control start
G50.5 P_ Q_ ; Composite control cancel

P: Composite axis 1 ID number
Q: Composite axis 2 ID number

G51.6 P_ Q_ ; Superimposed control start
G50.6 Q_ ; Superimposed control cancel

P: Superimposed master axis ID number
Q: Superimposed slave axis ID number

As an ID number, set a unique value for identifying each axis in parameter No. 12600 for both of P and Q.

G51.4/G50.4, G51.5/G50.5, and G51.6/G50.6 are one-shot G codes of group 00.

Explanation

Synchronous control

Synchronous control is performed with the G51.4/G50.4 commands, instead of simultaneously controlled axis selection signals.

Parameter setting examples for a 3-path system

- Parameter No.12600

	Path 1	Path 2	Path 3
X	101	201	301
Z	102	202	302

- Parameter No.8180

	Path 1	Path 2	Path 3
X	0	0	0
Z	0	102	102

- Program example (M100 to M103 are synchronization M codes.)

Path 1	Path 2	Path 3	Operation
N10 M100 P123 ;	M100 P123 ;	M100 P123 ;	Synchronization between paths 1, 2, and 3
N20 G51.4 P102 Q202 ; N30 G51.4 P102 Q302 ;			Start of Z1-Z2 synchronous control Start of Z1-Z3 synchronous control
N40 M101 P123 ;	M101 P123 ;	M101 P123 ;	Synchronization between paths 1, 2, and 3
N50 G00 Z100.;	Synchronous slave movement	Synchronous slave movement	Z1-Z2 and Z1-Z3 synchronous control
N60 M102 P123 ;	M102 P123 ;	M102 P123 ;	Synchronization between paths 1, 2, and 3
N70 G50.4 Q202 ; N80 G50.4 Q302 ;			Cancellation of Z1-Z2 synchronous control Cancellation of Z1-Z3 synchronous control
N90 M103 P123 ;	M103 P123 ;	M103 P123 ;	Synchronization between paths 1, 2, and 3

- Start of synchronous control

N20 G51.4 P102 Q202 :

Start of synchronous control with the Z1-axis being a master axis and the Z2-axis being a slave axis

N30 G51.4 P102 Q302 :

Start of synchronous control with the Z1-axis being a master axis and the Z3-axis being a slave axis

- Cancellation of synchronous control

N70 G50.4 Q202 :

Cancellation of synchronous control with the Z1-axis being a master axis and the Z2-axis being a slave axis

N80 G50.4 Q302 :

Cancellation of synchronous control with the Z1-axis being a master axis and the Z3-axis being a slave axis

- Parking

Use the L command to specify the start and cancellation of the parking of synchronous axes.
If the L command is omitted or if the L0 command is issued, the parking of both synchronous master axis and synchronous slave axis is canceled.

- Parameter check

If the axis number corresponding to the P specified with G51.4 is not set in slave axis parameter No. 8180, alarm PS5339, “ILLEGAL FORMAT COMMAND IS EXECUTED IN SYNC/MIX/OVL CONTROL.” is issued.

Composite control

Composite control is performed with the G51.5/G50.5 commands, instead of composite control axis selection signals.

Parameter setting examples for a 3-path system

- Parameter No.12600

	Path 1	Path 2	Path 3
X	101	201	301
Z	102	202	302

- Parameter No.8183

	Path 1	Path 2	Path 3
X	0	101	0
Z	0	102	0

- Program example (M100 to M103 are synchronization M codes.)

Path 1	Path 2	Path 3	Operation
N10 M100 P12 ;	M100 P12 ;	Independent operation	Synchronization between paths 1 and 2
N20 G51.5 P101 Q201 ; N30 G51.5 P102 Q202 ;			Start of X1-X2 composite control Start of Z1-Z2 composite control
N40 M101 P12 ;	M101 P12 ;		Synchronization between paths 1 and 2
N50 G00 X 100. Z100.;	Composite movement		X1-X2 and Z1-Z2 composite control
N60 M102 P12 ;	M102 P12 ;		Synchronization between paths 1 and 2
N70 G50.5 P101 Q201 ; N80 G50.5 P102 Q202 ;			Cancellation of X1-X2 composite control Cancellation of Z1-Z2 composite control
N90 M103 P12 ;	M103 P12 ;		Synchronization between paths 1 and 2

- Start of composite control

N20 G51.5 P101 Q201 :

Start of composite control on the X1- and X2-axes

N30 G51.5 P102 Q202 :

Start of composite control on the Z1- and Z2-axes

- Cancellation of composite control

N70 G50.5 P101 Q201 :

Cancellation of composite control on the X1- and X2-axes

N80 G50.5 P102 Q202 :

Cancellation of composite control on the Z1- and Z2-axes

- Parameter check

If the composite control axis corresponding to the P or Q specified with G51.5/G50.5 is not set in parameter No. 8183, alarm PS5339, “ILLEGAL FORMAT COMMAND IS EXECUTED IN SYNC/MIX/OVL CONTROL.” is issued.

Superimposed control

Superimposed control is performed with the G51.6/G50.6 commands, instead of superimposed control axis selection signals.

Parameter setting examples for a 3-path system

- Parameter No.12600

	Path 1	Path 2	Path 3
X	101	201	301
Z	102	202	302

- Parameter No.8186

	Path 1	Path 2	Path 3
X	0	0	0
Z	0	0	102

- Program example (M100 to M103 are synchronization M codes.)

Path 1	Path 2	Path 3	Operation
N10 M100 P13 ;	Independent operation	M100 P13 ;	Synchronization between paths 1 and 3
N20 G51.6 P102 Q302 ;			Start of Z1-Z3 superimposed control
N30 M101 P13 ;		M101 P13 ;	Synchronization between paths 1 and 3
N40 G00 Z100.;		G00 Z-200.;	Z1-Z3 superimposed control (Z+100 superimposed on Z3)
N50 M102 P13 ;		M102 P13 ;	Synchronization between paths 1 and 3
N60 G50.6 Q302 ;			Cancellation of Z1-Z3 superimposed control
N70 M103 P13 ;		M103 P13 ;	Synchronization between paths 1 and 3

- Start of superimposed control

N20 G51.6 P102 Q302 :

Start of superimposed control with the Z1-axis being a master axis and the Z3-axis being a slave axis

- Cancellation of superimposed control

N60 G50.6 Q302 :

Cancellation of superimposed control with the Z1-axis being a master axis and the Z3-axis being a slave axis

- Parameter check

If the axis number corresponding to the P specified with G51.6 is not set in superimposed slave axis parameter No. 8186, alarm PS5339, “ILLEGAL FORMAT COMMAND IS EXECUTED IN SYNC/MIX/OVL CONTROL.” is issued.

NOTE

1 If G codes (G50.4/G50.5/G50.6) for canceling synchronous, composite, and superimposed control with program commands are issued for axes under synchronous, composite, and superimposed control with DI signals, synchronous, composite, and superimposed control is canceled.

2 If the synchronous control axis selection signal, composite control axis selection signal, and superimposed control axis selection signal are changed from '1' to '0' for axes under synchronous, composite, and superimposed control with program commands, synchronous, composite, and superimposed control is canceled.

21.4 ROTARY AXIS ROLL-OVER

The roll-over function prevents coordinates for the rotary axis from overflowing. The roll-over function is enabled by setting bit 0 (ROAx) of parameter No. 1008 to 1.

21.4.1 Rotary Axis Roll-over

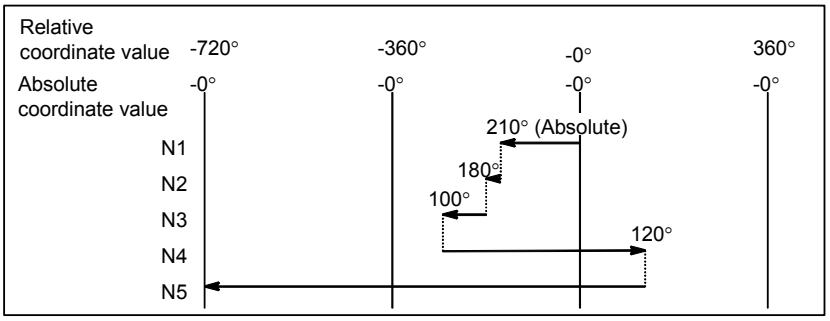
Explanation

For an incremental programming, the tool moves the angle specified in the command. For an absolute programming, the coordinates after the tool has moved are values set in parameter No. 1260, and rounded by the angle corresponding to one rotation. The tool moves in the direction in which the final coordinates are closest when bit 1 (RABx) of parameter No. 1008 is set to 0. Displayed values for relative coordinates are also rounded by the angle corresponding to one rotation when bit 2 (RRLx) of parameter No. 1008 is set to 1.

Example

Assume that axis A is the rotary axis and that the amount of movement per rotation is 360.000 (parameter No. 1260 = 360000). When the following program is executed using the roll-over function of the rotary axis, the axis moves as shown below.

G90 A0 ;	Sequence number	Actual movement value	Absolute coordinate value after movement end
N1 G90 A-150.0 ;	N1	-150	210
N2 G90 A540.0 ;	N2	-30	180
N3 G90 A-620.0 ;	N3	-80	100
N4 G91 A380.0 ;	N4	+380	120
N5 G91 A-840.0 ;	N5	-840	0



M

NOTE
This function cannot be used together with the index table indexing function.

21.4.2 Rotary Axis Control

This function controls a rotary axis as specified by an absolute command.
With this function, the sign of the value specified in the command is interpreted as the direction of rotation, and the absolute value of the specified value is interpreted as the coordinates of the target end position.

Explanation

This function is valid when rotary axis roll-over is enabled (bit 0 (ROAx) of parameter No. 1008 is set to 1).
If bit 3 (RAAx) of parameter No.1007 is set to 1, an absolute command specified for the roll-over rotary axis is interpreted as follows: The sign and absolute value of the value specified in the command represent the direction of rotation and the end position of movement respectively.
If bit 3 (RAAx) of parameter No.1007 is set to 0, the setting by bit 1 (RABx) of parameter No.1008 becomes significant.

NOTE
1 This function can be used only when the corresponding option is provided. (When a machining center system is used, this function cannot be used together with the index table indexing function.)
2 This function is valid for a roll-over rotary axis.
3 If bit 3 (RAAx) of parameter No.1007 is set to 1, bit 1 (RABx) of parameter No.1008 is ignored. To select a rotary motion of a shorter travel distance, set both RAAx and RABx to 0.

NOTE
4 This function is not supported when the machine coordinate system of the PMC axis control function is selected.

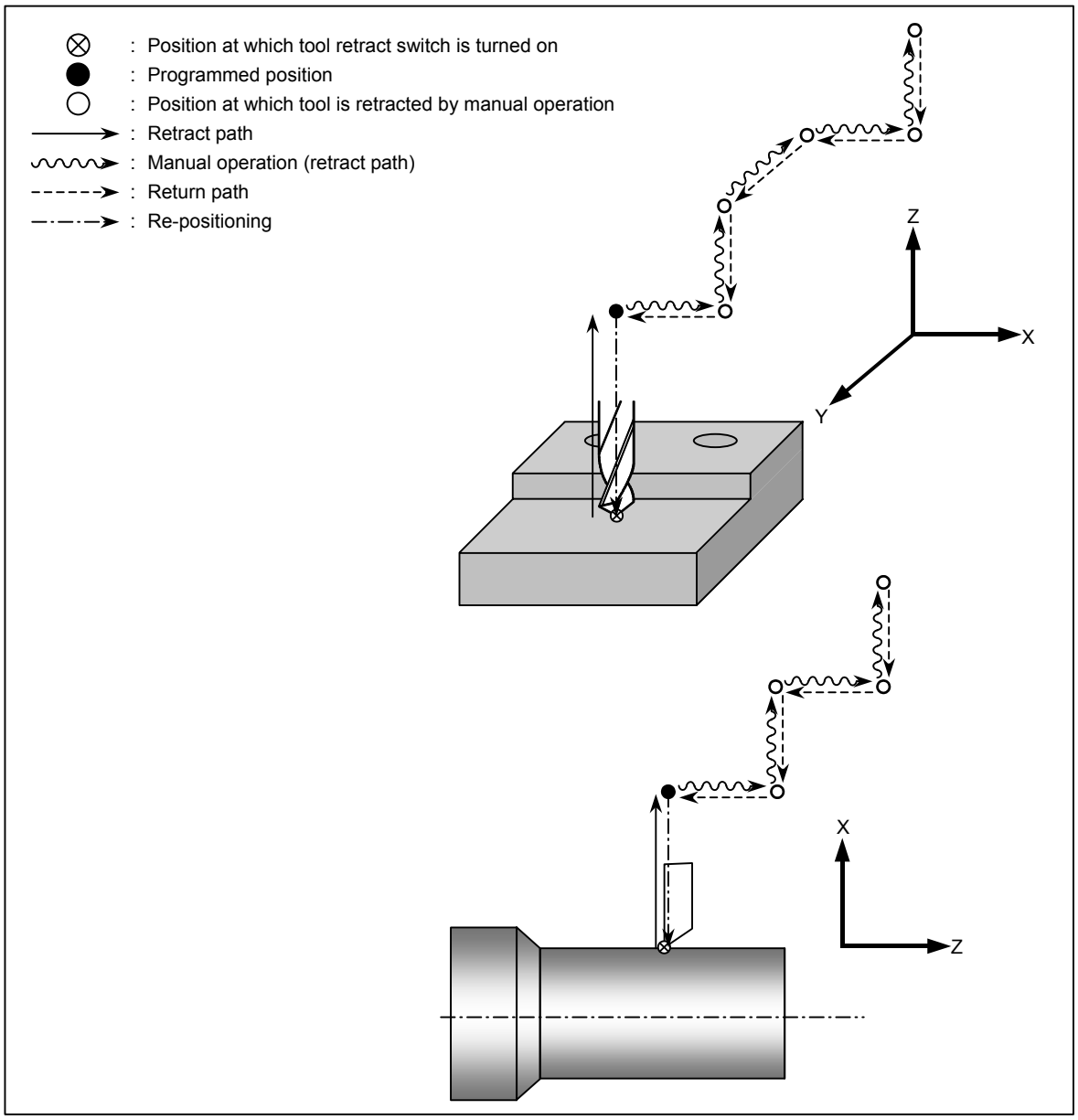
21.5 TOOL RETRACT AND RECOVER

Overview

To replace the tool damaged during machining or to check the status of machining, the tool can be withdrawn from a workpiece. The tool can then be advanced again to restart machining efficiently.
The tool retract and recover operation consists of the following four steps:

- Retract
  The tool is retracted to a predefined position using the TOOL WITHDRAW switch.
- Withdrawal
  The tool is moved to the tool-change position manually.
- Return
  The tool returns to the retract position.
- Repositioning
  The tool returns to the interrupted position.

For the tool retract and recover operations, see “Tool retract and recover “ in the Part III.



Format

Specify a retraction axis and distance in the following format:

Specify the amount of retraction, using G10.6.

G10.6 IP_ ;

IP: In incremental mode, retraction distance from the position where the retract signal is turned on
In the absolute mode, retraction distance to an absolute position
The specified amount of retraction is effective until G10.6 is next executed. To cancel the retraction, specify the following:
G10.6 ; (as a single block containing no other commands)

Explanation

- Retraction

When the TOOL WITHDRAW switch on the machine operator's panel is turned on during automatic operation or in the automatic operation stop or hold state, the tool is retracted the length of the programmed retraction distance. This operation is called retraction. The position at which retraction is completed is called the retraction position. The tool is retracted to the retraction position in linear interpolation at the minimum feedrate among the parameter (No.7042) values of moving axes. When retraction is completed, the RETRACTIONS POSITION LED on the machine operator's panel goes on.
When the TOOL WITHDRAW switch is turned on during execution of a block in automatic operation, execution of the block is interrupted immediately and the tool is retracted. After retraction is completed, the system enters the automatic operation hold state.
If the retraction distance and direction are not programmed, retraction is not performed. In this state, the tool can be withdrawn and returned.
When the TOOL WITHDRAW switch is turned on in the automatic operation stop or hold state, the tool is retracted, then the automatic operation stop or hold state is entered again.
When the TOOL WITHDRAW switch is turned on, the tool withdraw mode is set. When the tool withdraw mode is set, the TOOL BEING WITHDRAWN LED on the machine operator's panel goes on.

- Withdrawal

When the manual mode is set, the tool can be moved manually (jog feed, incremental feed, handle feed, or manual numeral command) to replace the tool or measure a machined workpiece. This operation is called a withdrawal. The tool withdrawal path is automatically memorized by the CNC.

- Return

When the mode is returned to automatic operation mode and the TOOL RETURN switch on the machine operator's panel is turned off, the CNC automatically moves the tool to the retraction position by tracing the manually-moved tool path backwards. This operation is called a return. Upon completion of a return to the retraction position, the RETRACTIONS POSITION LED comes on. The return is performed at the feedrate specified by parameter (No.7042) for each axis.

- Repositioning

When the cycle start button is pressed while the tool is in the retraction position, the tool moves to the position where the TOOL WITHDRAW switch was turned on. This operation is called repositioning. The re-positioning movement is made by linear interpolation at the minimum feedrate among the parameter (No.7042) values of moving axes. Upon completion of repositioning, the TOOL BEING WITHDRAWN LED is turned off, indicating that the tool withdrawal mode has terminated. Operation after completion of repositioning depends on the automatic operation state when the tool withdrawal mode is set.

(1) When the tool withdrawal mode is set during automatic operation, operation is resumed after completion of repositioning.
(2) When the tool withdrawal mode is set when automatic operation is held or stopped, the original automatic operation hold or stop state is set after completion of repositioning. When the cycle start button is pressed again, automatic operation is resumed.

Limitation

- Offset

If the origin, presetting, or workpiece origin offset value (or External workpiece origin offset value) is changed after retraction is specified with G10.6 in absolute mode, the change is not reflected in the retraction position. After such changes are made, the retraction position must be respecified with G10.6.
When the tool is damaged, automatic operation can be interrupted with a tool withdrawal and return operation in order to replace the tool. Note that if the tool offset value is changed after tool replacement, the change is ignored when automatic operation is resumed from the start point or other point in the interrupted block.

- Machine lock, mirror image, and scaling

When withdrawing the tool manually in the tool withdrawal mode, never use the machine lock, mirror-image, or scaling function.

- Reset

Upon reset, the retraction data specified in G10.6 is cleared. Retraction data needs to be specified again.

- Retraction command

The tool withdrawal and return function is enabled even when the retraction command is not specified. In this case, retraction and repositioning are not performed.

> ⚠ **WARNING**
> The retraction axis and retraction distance specified in G10.6 need to be changed in an appropriate block according to the figure being machined. Be very careful when specifying the retraction distance; an incorrect retraction distance may damage the workpiece, machine, or tool.

21.5.1 Improvement of Tool compensation for Tool Retract and Recover

Outline

In this function, when the recovery operation or re-positioning operation is started, the updated compensation value is used. Therefore the restart operation is performed with the updated compensation value.

Explanation

If this function is effective (the bit 7 (TRO) of parameter No.7002 is set to 1), when the recovery operation or re-positioning operation is started, the updated compensation value is used and the restart operation is performed with the updated compensation value. Thus, if the compensation value is updated after exchanging the tool, the tip point of tool becomes the same position as it of before exchanging tool in the restart operation thereafter.

The operation that updated compensation value is effective can be selected by the bit 6 (TNR) of parameter No.7002.
Example-1 (When the bit 6 (TNR) of parameter No.7002 is set to 0):
Assuming that the compensation value is updated from OFS_1 to OFS_2 at the manual retract position. When the recovery operation is started, the updated compensation value OFS_2 is used, the tip point of new tool and old tool is the same position in the retract position. (Refer to the figure 1)

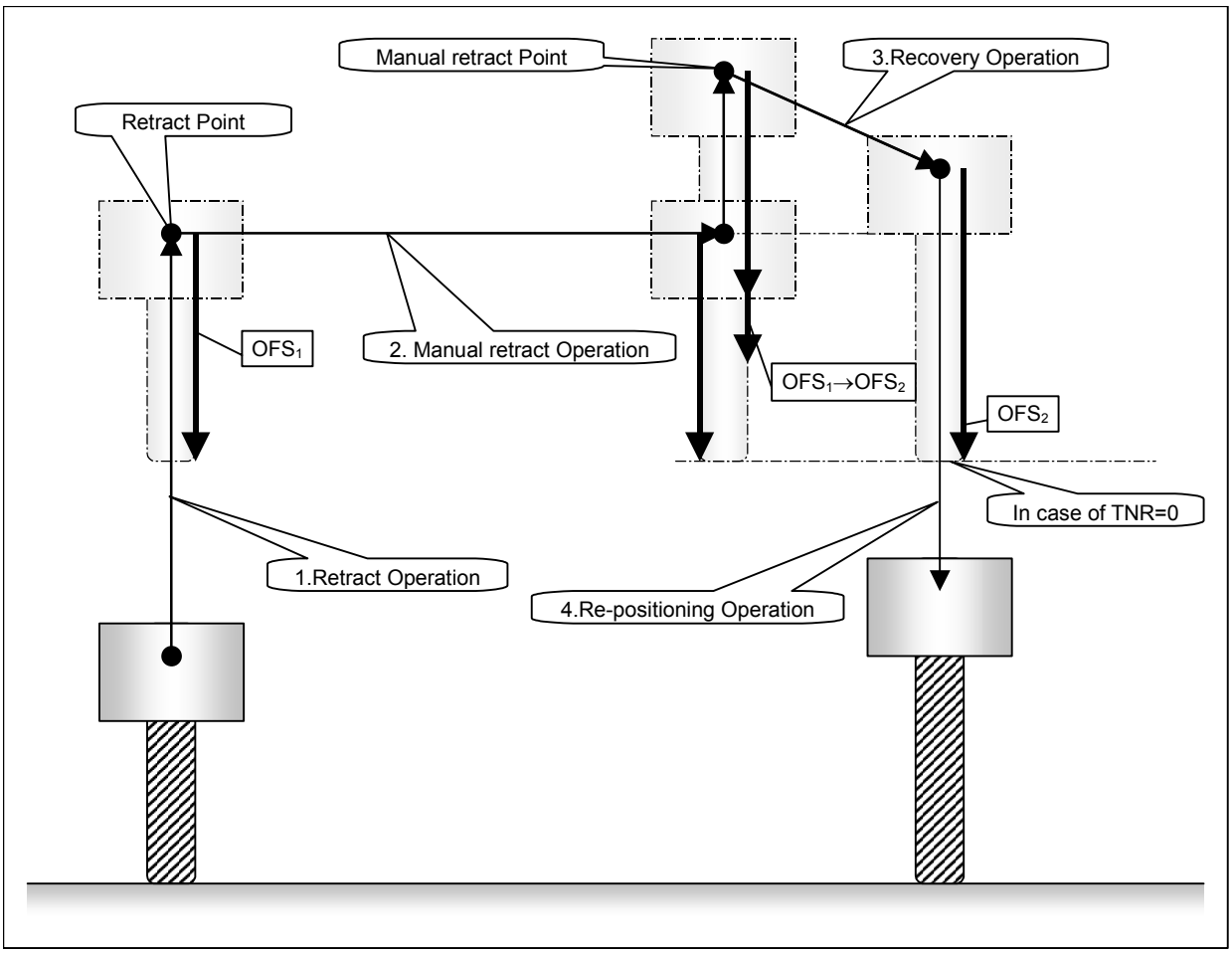

Manual retract Point
3.Recovery Operation
Retract Point
OFS1
2. Manual retract Operation
OFS1→OFS2
OFS2
In case of TNR=0
1.Retract Operation
4.Re-positioning Operation

Fig. 21.5.1 (a)

Example-2 (When the bit 6 (TNR) of parameter No.7002 is set to 1):
Assuming that the compensation value is updated from OFS_1 to OFS_2 in the manual retract position. When the recovery operation is started, the updated compensation value OFS_2 is not used. After re-positioning operation, the updated compensation value OFS_2 is used (Refer to the figure 2)

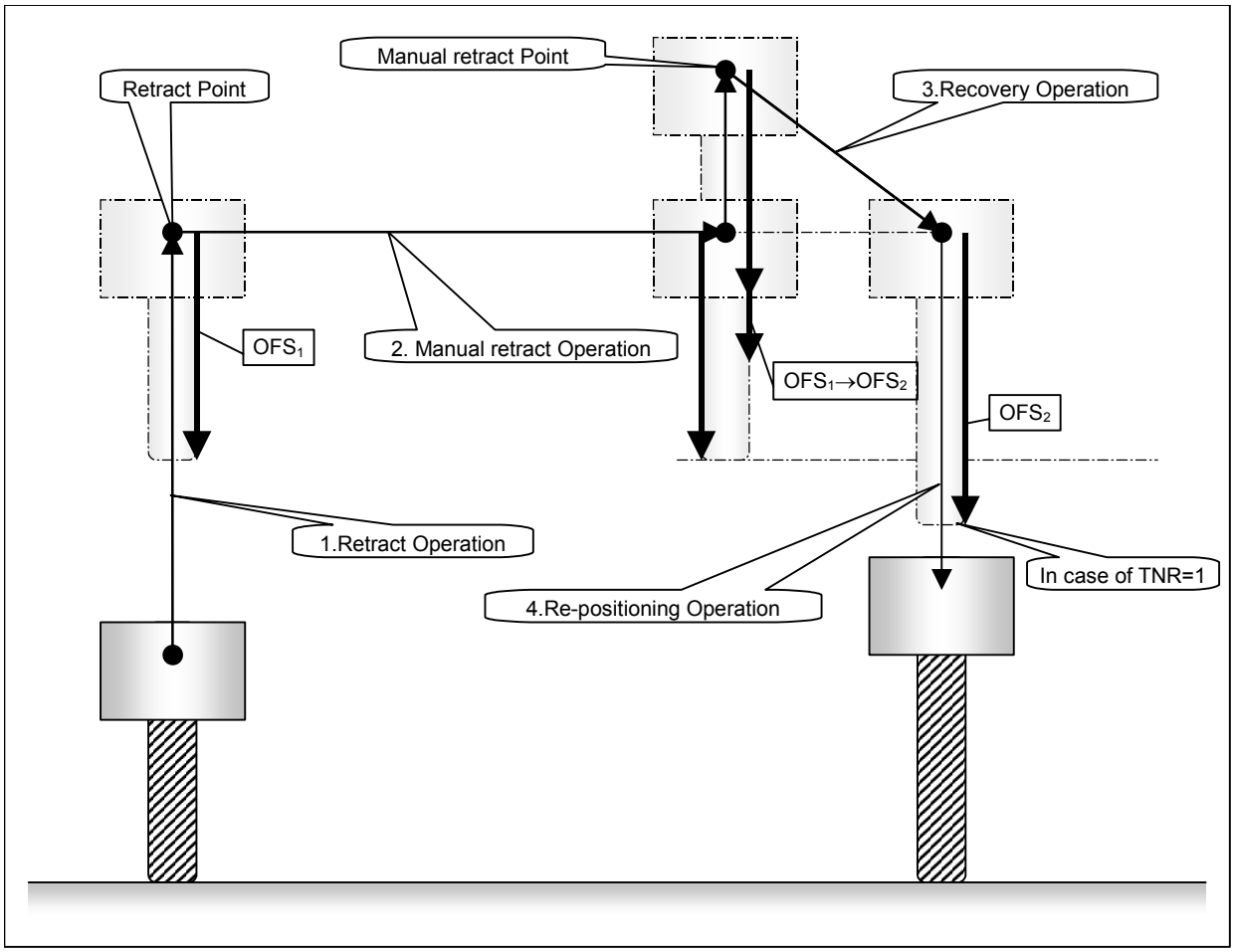


Fig. 21.5.1 (b)

Notes

NOTE

1 The compensation considered by this function is as follows. Neither the cutter compensation nor the tool nose radius compensation is considered.
Machining system : Tool length compensation G43/G44 (wear/geometry),
Tool Offset G43.7/G44.7 (wear /geometry)
Lathe system : When the Extended Tool Selection is not used (the bit 3 (TCT) of parameter No.5040 is set to 0)
Tool compensation T code (wear /geometry)
When the Extended Tool Selection is not used (the bit 3 (TCT) of parameter No.5040 is set to 1)
Tool length compensation G43/G44 (wear /geometry),
Tool Offset G43.7/G44.7 (wear /geometry)

2 The compensation value can be changed only in the retract position or the manual retract position.

NOTE

3 When the tool length compensation B is used, the vertical axis against the plane from which the tool retract and recover is executed is compensated. This plane is not a same plane when G43/G44 is instructed (In case machining system)
4 When the tool length compensation C is used and G43 or G44 is commanded, the compensated axis is indefinite. Therefore this function cannot be used in this case.
5 This function cannot be used during the movement to the reference position by the G28/G30 command or the G53 command is executing.
6 When the Tool center point control is used, this function cannot be used.

21.6 ELECTRONIC GEAR BOX

21.6.1 Electronic Gear Box

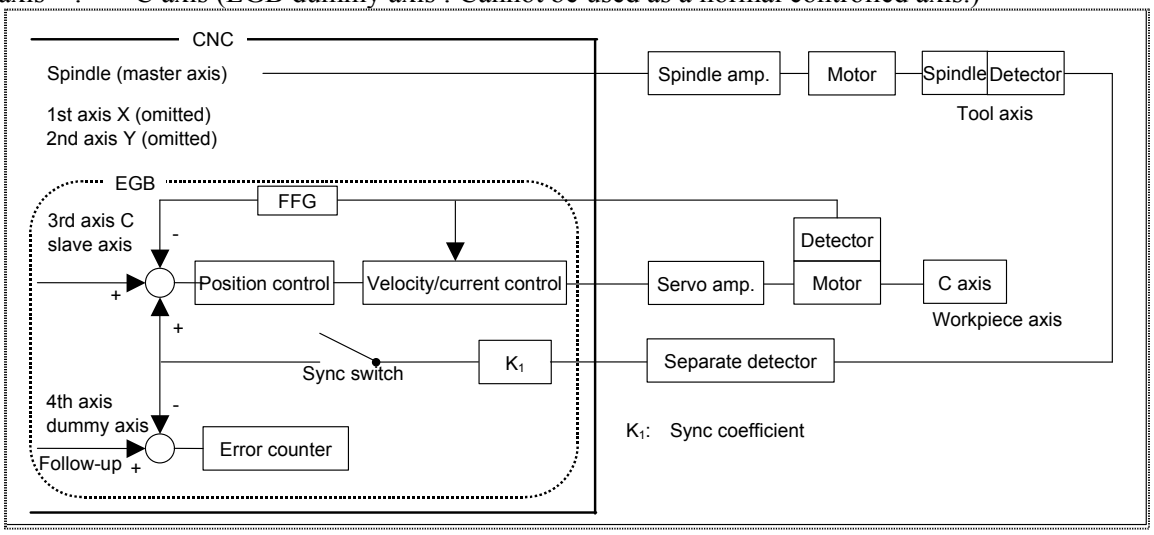
Overview

This function enables fabrication of high-precision gears, screws, and other components by rotating the workpiece in synchronization with a rotating tool or by moving the tool in synchronization with a rotating workpiece. The rate of synchronization can be specified with a program. The synchronization of tool and workpiece axes with this function adopts a system in which the synchronization is directly controlled by digital servo, so that the workpiece axis can follow up the speed fluctuations on the tool axis with no error, thereby allowing fabrication of high-precision cogwheels. In the subsequent explanation, the Electronic Gear Box is called the EGB.

There are several conditions in the setting and so on of the workpiece axis and the tool axis, and refer to the appropriate manual provided by the machine tool builder.

- Example of controlled axis configuration

Spindle : EGB master axis : Tool axis
1st axis : X axis
2nd axis : Y axis
3rd axis : C axis (EGB slave axis : Workpiece axis)
4th axis : C axis (EGB dummy axis : Cannot be used as a normal controlled axis.)



Format

	Parameter EFX(No.7731#0)=0	Parameter EFX(No.7731#0)=1	
		Parameter HBR(No.7731#5)=1	Parameter HBR(No.7731#5)=0
Start of synchronization	**G81 T__ (L__) (Q__ P__) ;**	**G81.4 R__ (L__) (Q__ P__) ;**	**G81.4 T__ (L__) (Q__ P__) ;**
Cancellation of synchronization	**G80 ;**	**G80.4 ;**	**G80.4 ;**
	(*1) (*4)	**(*2) (*4)**	**(*3) (*4)**

T(or R) : Number of teeth (Specifiable range: 1 to 5000)
L : Number of hob threads (Specifiable range: -250 to +250)
The sign of L determines the direction of rotation for the workpiece axis.
When L is positive, the direction of rotation for the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation for the workpiece axis is negative (- direction).
When L is 0, it follows the setting of bit 3 (LZR) of parameter No. 7701.
If L is not specified, the number of hob threads is assumed 1.
Q : Module or diametral pitch
Specify a module in the case of metric input.
(Unit: 0.00001mm, Specifiable range: 0.01 to 25.0mm)
Specify a diametral pitch in the case of inch input.
(Unit: 0.00001inch^{-1}, Specifiable range: 0.01 to 254.0 inch^{-1})
P : Gear helix angle
(Unit: 0.0001deg, Specifiable range: -90.0 to +90.0deg)
*1 Use it for machining centers.
*2 Use it for lathes.
*3 Use it for machining centers.
This format enables specification of the same G codes as those for lathes.
*4 When specifying Q and P, the user can use a decimal point.

NOTE
Specify G81, G80, G81.4, and G80.4 in a single block.

Explanation

- Master axis, slave axis, and dummy axis

The synchronization reference axis is called the master axis, while the axis along which movement is performed in synchronization with the master axis is called the slave axis. For example, if the workpiece moves in synchronization with the rotating tool as in a hobbing machine, the tool axis is the master axis and the workpiece axis is the slave axis.
Which axes to become the master and slave axes depends on the configuration of the machine. For details, refer to the manual issued by the machine tool builder.
A single servo axis is used exclusively so that digital servo can directly read the rotation position of the master axis. (This axis is called the EGB dummy axis.)

- Synchronous control

(1) Start of synchronization

If G81 is issued so that the machine enters synchronization mode, the synchronization switch of the EGB function is closed, and the synchronization of the tool and workpiece axes is started. During synchronization, the rotation about the tool and workpiece axes is controlled so that the relationship between T (number of teeth) and L (number of hob threads) is maintained. During synchronization, the synchronization relationship is maintained regardless of whether the operation is automatic or manual.

Specify P and Q to use helical gear compensation.

If only either P or Q is issued, alarm PS1594 is generated.

If, during synchronization, G81 is issued again without synchronization cancelation, alarm PS1595 is generated if ECN, bit 3 of parameter No. 7731, is 0. If ECN, bit 3 of parameter No. 7731, is 1, helical gear compensation is conducted with the synchronization coefficient being changed to the one newly specified with T and L commands if T and L commands are issued, and if T and L commands are not issued and only P and Q commands are issued, helical gear compensation is conducted with the synchronization coefficient kept intact. This allows consecutive fabrication of helical gears and super gears.

(2) Start of tool axis rotation

When the rotation of the tool axis starts, the rotation of the workpiece axis starts so that the synchronous relationship specified in the G81 block can be maintained.

The rotation direction of the workpiece axis depends on the rotation direction of the tool axis. That is, when the rotation direction of the tool axis is positive, the rotation direction of the workpiece axis is also positive; when the rotation direction of the tool axis is negative, the rotation direction of the workpiece axis is also negative. However, by specifying a negative value for L, the rotation direction of the workpiece axis can be made opposite to the rotation direction of the tool axis.

During synchronization, the machine coordinates of the workpiece axis and EGB axis are updated as synchronous motion proceeds. On the other hand, a synchronous move command has no effect on the absolute and relative coordinates.

(3) Termination of tool axis rotation

Synchronizing with gradual stop of the tool axis, the workpiece axis is decelerated and stopped. By specifying the G80 command after the spindle stops, synchronization is canceled, and the EGB synchronization switch is opened.

(4) Cancellation of synchronization

When cancellation of synchronization is issued, the absolute coordinate on the workpiece axis is updated in accordance with the amount of travel during synchronization. Subsequently, absolute commands for the workpiece axis will be enabled.

For a rotation axis, the amount of travel during synchronization, as rounded to 360-degree units is added to the absolute coordinate.

In the G80 block, only O and N addresses can be specified.

By setting bit 0 (HBR) of parameter No. 7700 to 0, it is possible to cancel synchronization with a reset.

Synchronization is automatically canceled under the following conditions:

<1> An emergency stop is applied.
<2> A servo alarm is generated.
<3> Alarm PW0000, “POWER MUST BE OFF” is generated.
<4> An IO alarm is generated.

⚠ CAUTION

1 Feed hold, interlock, and machine lock are invalid to a slave axis in EGB synchronization.

⚠ CAUTION

2 Even if an OT alarm is issued for a slave axis in EGB synchronization, synchronization will not be canceled.
3 During synchronization, it is possible to execute a move command for a slave axis and other axes, using a program. The move command for a slave command must be an incremental one.

NOTE

1 If bit 0 (HBR) of parameter No. 7700 is set to 1, EGB synchronization will not be canceled due to a reset. Usually, set this parameter bit to 1.
2 In synchronous mode, it is not possible to specify G27, G28, G29, G30, G30.1, and G53 for a slave axis.
3 It is not possible to use controlled axis detach for a slave axis.
4 During synchronization, manual handle interruption can be performed on the slave and other axes.
5 In synchronization mode, no inch/metric conversion commands (G20 and G21) cannot be issued.
6 In synchronous mode, only the machine coordinates on a slave axis are updated.
7 If bit 0 (EFX) of parameter No. 7731 is 0, no canned cycle for drilling can be used. To use a canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.
8 If TDP, bit 0 of parameter No. 7702, is 1, the permissible range of T is 0.1 to 100 (1/10 of the specified value).
9 If, at the start of EGB synchronization (G81), L is specified as 0, synchronization starts with L assumed to be 1 if bit 3 (LZR) of parameter No. 7701 is 0; if bit 3 (LZR) of parameter No. 7701 is 1, synchronization is not started with L assumed to be 0. At this time, helical gear compensation is performed.
10 Feed per revolution is performed on the feedback pulses on the spindle. By setting of bit 0 (ERV) of parameter No. 7703, to 1, feed per revolution can be performed based on the speed on the synchronous slave axis.
11 Actual cutting feedrate display does not take synchronization pulses into consideration.
12 For an EGB slave axis, synchronous and composite control cannot be executed.
13 In EGB synchronization mode, AI contour control mode is temporarily canceled.

- Helical gear compensation

For a helical gear, the workpiece axis is compensated for the movement along the Z-axis (axial feed axis) based on the torsion angle of the gear.
Helical gear compensation is performed with the following formulas:

$$\text{Compensation angle} = \frac{Z \times \sin(P)}{\pi \times T \times Q} \times 360 \text{ (for metric input)}$$

$$\text{Compensation angle} = \frac{Z \times Q \times \sin(P)}{\pi \times T} \times 360 \text{ (for inch input)}$$

where

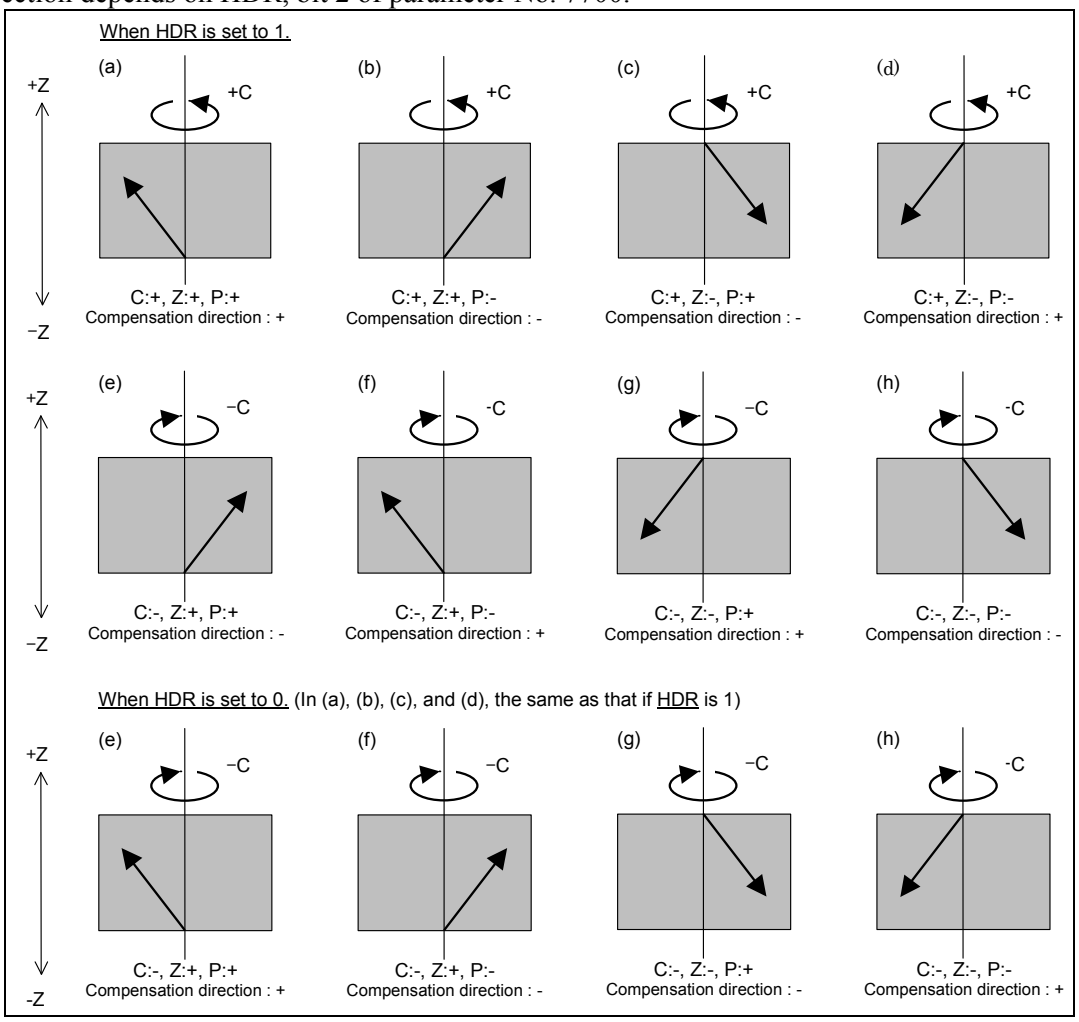
Compensation angle: Signed absolute value (deg)
Z : Amount of travel on the Z-axis after the specification of G81
P : Signed gear helix angle (deg)
π : Circular constant

T : Number of teeth
Q : Module (mm) or diametral pitch ($inch^{-1}$)
Use P, T, and Q specified in the G81 block.

In helical gear compensation, the machine coordinates on the workpiece axis and the absolute coordinates are updated with helical gear compensation.

- Direction of helical gear compensation

The direction depends on HDR, bit 2 of parameter No. 7700.



- Synchronization coefficient

A synchronization coefficient is internally represented using a fraction (Kn/Kd) to eliminate an error. The formula below is used for calculation.

$$\text{Synchronization coefficient} = \frac{K_n}{K_d} = \frac{L}{T} \times \frac{\beta}{\alpha}$$

where
L : Number of hob threads
T : Number of teeth
α : Number of pulses of the position detector per rotation about the master axis (parameter No. 7772)
β : Number of pulses of the position detector per rotation about the slave axis (parameter No. 7773)

Kn / Kd is a value resulting from reducing the right side of the above formula, but the result of reduction is subject to the following restrictions:
$-2147483648 \leq K_n \leq 2147483647$
$1 \leq K_d \leq 2147483647$
When this restriction is not satisfied, the alarm PS1596 is issued when G81 is specified.

Example

```
O1000   ;
N0010   M19   ;              Tool axis orientation
N0020   G28   G91   C0   ;  Reference position return on the workpiece axis
N0030   G81   T20   L1   ;  Synchronous start on tool and workpiece axes
                             (Rotation about the workpiece axis by 18° per rotation about the tool axis)
N0040   S300   M03   ;       Rotation about the tool axis at 300min-1
N0050   G01 X__F__;          Movement along the X-axis (cut)
N0060   G01 Z__F__;          Movement along the Z-axis (machining)
-------------------------- ; If necessary, axis commands such as C, X, and Z commands are allowed.
N0100   G01 X__F__;          Movement along the X-axis (escape)
N0110   M05   ;              Stop on the tool axis
N0120   G80   ;              Synchronous cancellation on tool and workpiece axes
N0130   M30   ;
```

- Retract function

(1) Retract function with an external signal
When the retract switch on the machine operator's panel is turned on, retraction is performed with the retract amount set in parameter No. 7741 and the feedrate set in parameter No. 7740. No movement is performed along an axis for which 0 is set as the retract amount.
For the retract switch, refer to the relevant manual provided by the machine tool builder.

(2) Retract function with an alarm
If, during EGB synchronization or automatic operation, a CNC alarm is issued, retraction is performed with the retract amount set in parameter No. 7741 and the speed set in parameter No. 7740.
This can prevent the tool and the object being machined from damage if a servo alarm is generated.
No movement is performed along an axis for which 0 is set as the retract amount.
For the retract switch, refer to the relevant manual provided by the machine tool builder.

Conditions under the retract function with an alarm
The conditions under which the retract function is performed by an alarm can be changed using the settings of bit 1(ARE) of parameter No. 7703, and bit 2 (ARO) of parameter No. 7703.
The table below lists parameter settings and corresponding conditions.

ARE	ARO	Condition
1	0	EGB synchronization is in progress.
1	1	Both EGB synchronization and automatic operation are in progress.
0	0	Either EGB synchronization or automatic operation is in progress.
0	1	

⚠ CAUTION
1 Retraction is performed at the speed specified in parameter No. 7740.
2 Feed hold is not effective to movement during retraction.
3 Feedrate override is not effective to movement during retraction.

NOTE
1 During a retract operation, an interlock is effective to the retract axis.
2 During a retract operation, a machine lock is effective to the retract axis.
3 The retraction direction depends on the movement direction of the machine, regardless of whether an mirror image (signal and setting) is enabled or disabled. (No mirror image can be applied to the updating of absolute coordinates.)
4 If retraction is performed during automatic operation, automatic operation is halted simultaneously with a retract operation, but it is at the end of the retract operation that the operation state switches to the automatic operation halt state.
5 It is not possible to perform automatic operation during retraction.
6 The acceleration/deceleration of a retract operation is in the acceleration/deceleration state at the start of retraction.
7 Retract movement is performed with non-linear type positioning.
8 If, during a retract operation, a reset or an emergency stop is made, the operation is interrupted.
9 To enable the retract function with an alarm, bit 3 (ART) of parameter No. 7702, must be set.
10 The retract function with an alarm does not perform a retract operation on the retract axis if an overtravel alarm or a servo alarm is generated on the retract axis.
11 If a new alarm is issued during retraction with the retract function with an alarm, a retract operation is not performed.

21.6.2 Spindle Electronic Gear Box

Overview

In the system with two spindles for a tool axis and a workpiece axis, a gear can be shaped (grind/cut) by synchronizing the workpiece axis rotation with the tool axis (grinding axis /hob) rotation.
To synchronize these two spindles, the spindle electronic gear box is used (Hereafter, the electronic gear box is called the EGB.). In the spindle EGB, the synchronous pulse is produced from the feedback pulse of the position detector attached to the tool axis (master axis), and the workpiece axis (slave axis) rotates with the synchronous pulse. Feedback pulse from master side to slave side is sent by the communication between spindle amplifiers.

Specification

The specifications of the spindle EGB control are as follows:
(1) The Spindle EGB synchronization is started by specifying T command (number of teeth) and L command (number of hob threads), which determine the synchronous ratio, in G81 block. The Spindle EGB synchronization is canceled by specifying G80.
(2) The synchronous ratio is calculated from T and L command in G81 block and the number of position detector pulses per rotation about the tool and the workpiece axis (parameter setting).
(3) This function has the retract function.
(4) The cutting helical gear is performed by specifying Q command (module or diametral pitch) and P command (gear helix angle) in G81 block.
(5) The Spindle EGB synchronization is maintained regardless of whether the operation is automatic or manual.
(6) In order to turn on to spindle EGB synchronization mode, the slave spindle has to be Cs contouring control mode, though the master spindle may be any mode.

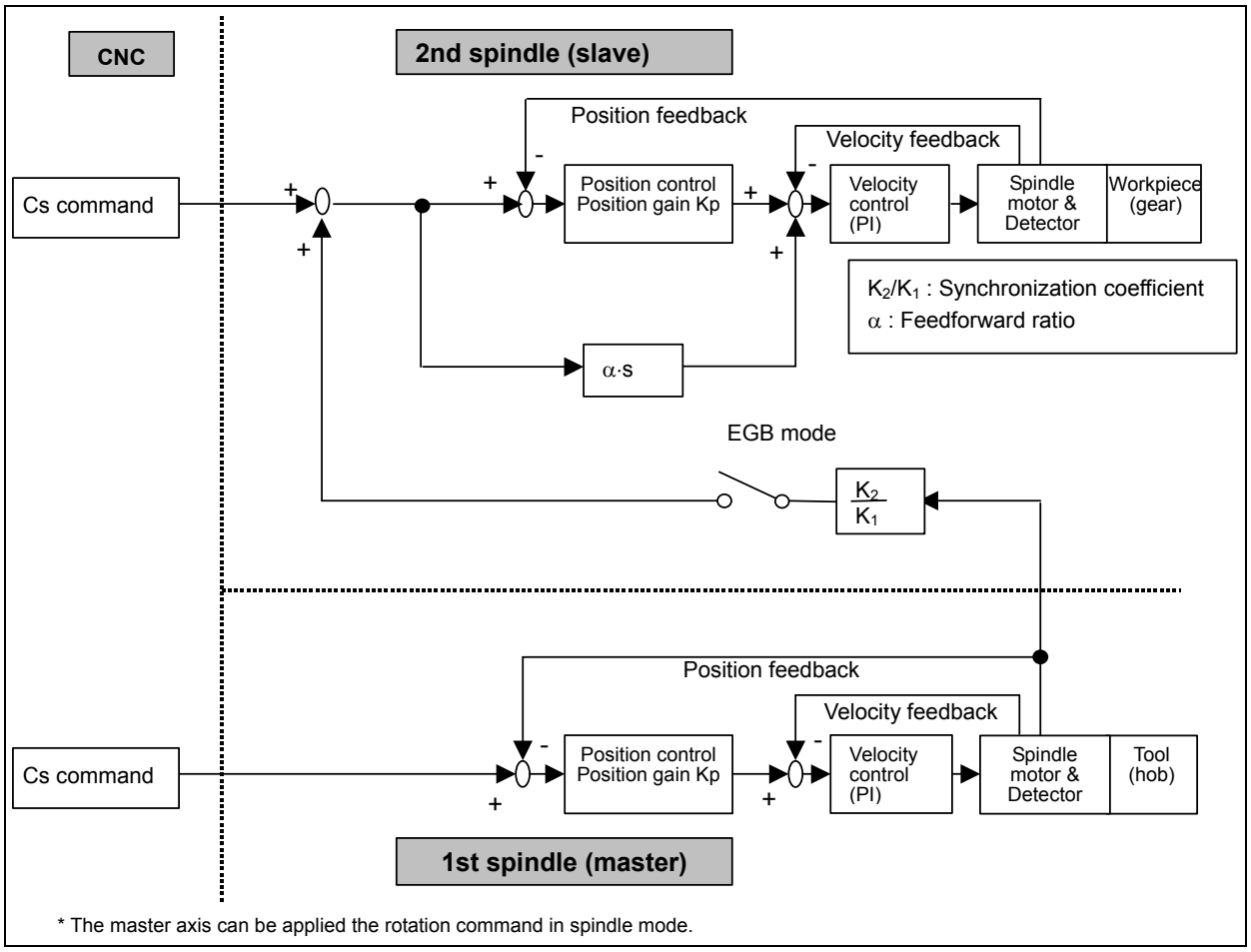


Fig. 21.6.2 (a) Block diagram of spindle EGB control

Format

	Parameter EFX (No. 7731#0)=0	Parameter EFX(No. 7731#0)=1	
		Parameter HBR (No. 7731#5)=1	Parameter HBR (No. 7731#5)=0
Start of synchronization	**G81 T__ (L__) (Q__ P__) ;**	**G81.4 R__ (L__) (Q__ P__) ;**	**G81.4 T__ (L__) (Q__ P__) ;**
Cancellation of synchronization	**G80 ;**	**G80.4 ;**	**G80.4 ;**
	(*1) (*4)	**(*2) (*4)**	**(*3) (*4)**

T(or R) : Number of teeth (Specifiable range: 1 to 5000)
L : Number of hob threads (Specifiable range: -250 to +250)
The sign of L determines the direction of rotation for the workpiece axis.
When L is positive, the direction of rotation about the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation about the workpiece axis is negative (- direction).
When L is 0, it depends on the setting of bit 3 (LZR) of parameter No. 7701.
If L is not specified, the number of hob threads is assumed 1.
Q : Module or diametral pitch
Specify a module in the case of metric input.
(Unit: 0.00001mm, Specifiable range: 0.01 to 25.0mm)
Specify a diametral pitch in the case of inch input.
(Unit: 0.00001inch-1, Specifiable range: 0.01 to 254.0 inch-1)
P : Gear helix angle
(Unit: 0.0001deg, Specifiable range: -90.0 to 90.0deg)
*1 Use it for machining centers.
*2 Use it for lathes.
*3 Use it for machining centers.
This format enables specification of the same G codes as those for lathes.
*4 When specifying Q and P, the user can use a decimal point.

NOTE
Specify G81, G80, G81.4, and G80.4 singly in a block.

Explanation

- Parameter setting

The following parameters should be set for the Spindle EGB control.
(1) Slave axis number (Parameter No. 7710)
(2) Number of position detector pulses per rotation about the tool axis (Parameter No. 7772)
(3) Number of position detector pulses per rotation about the workpiece axis (Parameter No. 7773)
(4) Spindle EGB master axis enable (Parameter No. 4352#7)
(5) Spindle EGB slave axis enable (Parameter No. 4352#6)
(6) Number of sinusoidal waves from master spindle position detector(Parameter No. 4386)

- Start/Cancellation of synchronization

When the rotation of the tool axis (master axis) starts after G81 is specified, the EGB synchronization starts with the synchronous relationship specified in G81 block, and the rotation of the workpiece axis (slave axis) starts. When the EGB synchronization starts, the synchronization switch of the EGB function is closed.

When the rotation of the tool axis is stopped, the workpiece axis stopped with keeping synchronization. Then the EGB synchronization is canceled by specifying G80. When the EGB synchronization is canceled, the EGB synchronization switch is opened.
Specify P and Q to use helical gear compensation.
If only either P or Q is issued, alarm PS1594 is generated.
G81 cannot be specified again during the EGB synchronization. Moreover, the specification of T, L, Q, and P cannot be modified during the synchronization. Specify the starting and canceling of synchronization during the tool axis (master axis) stopping.

⚠ CAUTION
1 Feed hold, interlock, and machine lock are invalid to a slave axis in EGB synchronization.
2 Even if an OT alarm is issued for a slave axis in EGB synchronization, synchronization will not be canceled.
3 During synchronization, it is possible to execute a move command for a slave axis and other axes, using a program. The move command for a slave command must be an incremental one.

NOTE
1 If bit 0 (HBR) of parameter No. 7700 is set to 1, EGB synchronization will not be canceled due to a reset. Usually, set this parameter bit to 1.
2 The EGB synchronization should be started and canceled during the stop of both the master and the slave axis.
3 Reference position return of Cs contouring control axis for master and slave axis should be performed before G81 is specified. In the synchronization mode, reference position return cannot be performed. Do not change to reference position return mode in the synchronization mode.
4 In case that the parameters for axis setting (No. 7710, 4352) are not set correctly, alarm PS1593 occurs when G81 specified.
5 In synchronous mode, it is not possible to specify G27, G28, G29, G30, G30.1, and G53 for a slave axis.
6 In EGB synchronization mode, AI contour control mode is temporarily canceled.
7 The slave axis position display is updated by synchronous pulse as follows :
- In synchronous mode, only the machine coordinates on a slave axis are updated. Absolute coordinates and relative coordinates are not updated.
- When the synchronization is canceled, the pulses of the synchronization rounded to 360-degree units are added to the absolute coordinates.

8 The rotation direction of the slave axis depends on the rotation direction of the master axis. That is, when the rotation direction of the master axis is positive, the rotation direction of the slave axis is also positive; when the rotation direction of the master axis is negative, the rotation direction of the slave axis is also negative. However, by specifying a negative value for L, the rotation direction of the slave axis can be made opposite to the rotation direction of the master axis.
9 The synchronization mode is canceled by servo alarm, spindle alarm, PW alarm, or emergency stop.
10 The synchronization is not maintained in servo off status of the slave axis.
11 During synchronization, manual handle interruption can be performed on the slave and other axes.
12 In synchronization mode, no inch/metric conversion commands (G20 and G21) can be issued.

NOTE

13 If bit 0 (EFX) of parameter No. 7731 is 0, no canned cycle for drilling can be used. To use a canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.
14 If TDP, bit 0 of parameter No. 7702, is 1, the permissible range of T is 0.1 to 100 (1/10 of the specified value).
15 If, at the start of EGB synchronization (G81), L is specified as 0, synchronization starts with L assumed to be 1 if bit 3 (LZR) of parameter No. 7701, is 0; if bit 3 (LZR) of parameter No. 7701, is 1, synchronization is not started with L assumed to be 0. At this time, helical gear compensation is performed.
16 Feed per revolution is performed on the feedback pulses on the spindle. By setting ERV, bit 0 of parameter No. 7703, to 1, feed per revolution can be performed based on the speed on the synchronous slave axis.
17 Actual cutting feedrate display does not take synchronization pulses into consideration.
18 For an EGB slave axis, synchronous and composite control cannot be executed.
19 Spindle EGB cannot be used by the G81 command together with servo EGB. Please use the G81 command for spindle EGB and G81.5 command for servo EGB when using it together.
20 When simple spindle EGB function is used, this function cannot be used. When G81 is commanded for slave axis of simple spindle EGB, PS1593 alarm is generated.
21 The master axis cannot be shared by simple spindle EGB function and this function.

- Program example

Axis configuration: X,Y,Z, B(Cs contouring control axis : tool axis/master axis), C(Cs contouring control axis : workpiece axis/slave axis)

```
O1000 ;
N0010 G80 ;
N0020 G28 G91 B0 C0 ;   Reference position return operation on the tool and workpiece axis.
N0030 G81 T20 L1 ;      Start of synchronization.
N0040 Mxx ;             Rotate the tool axis by velocity command of PMC axis control.
N0050 G04 X1000 ;       Wait until rotation of the tool axis is constant.
N0060 G01 X_ F_ ;       Movement along the X-axis(cutting).
N0070 G01 Z_ F_ ;       Movement along the Z-axis(machining).
-------------------------
-------------------------
N0100 G01 X_ F_ ;       Movement along the X-axis(escape).
N0110 M05 ;             Stop the tool axis.
N0120 G80 ;             Cancel synchronization.
N0130 M30 ;
```

- Helical gear compensation

For a helical gear, the workpiece axis is compensated for the movement along the Z-axis (axial feed axis) based on the torsion angle of the gear.
Helical gear compensation is performed with the following formulas:

$$\text{Compensation angle} = \frac{Z \times \sin(P)}{\pi \times T \times Q} \times 360 \text{ (for metric input)}$$

$$\text{Compensation angle} = \frac{Z \times Q \times \sin(P)}{\pi \times T} \times 360 \text{ (for inch input)}$$

where

Compensation angle: Signed absolute value (deg)

Z : Amount of travel on the Z-axis after the specification of G81

P : Signed gear helix angle (deg)

π : Circular constant

T : Number of teeth

Q : Module (mm) or diametral pitch ($inch^{-1}$)

Use P, T, and Q specified in the G81 block.

In helical compensation, the machine coordinates and the absolute coordinates on the workpiece axis are updated with helical compensation.

- Direction of helical gear compensation

The direction depends on HDR, bit 2 of parameter No. 7700.

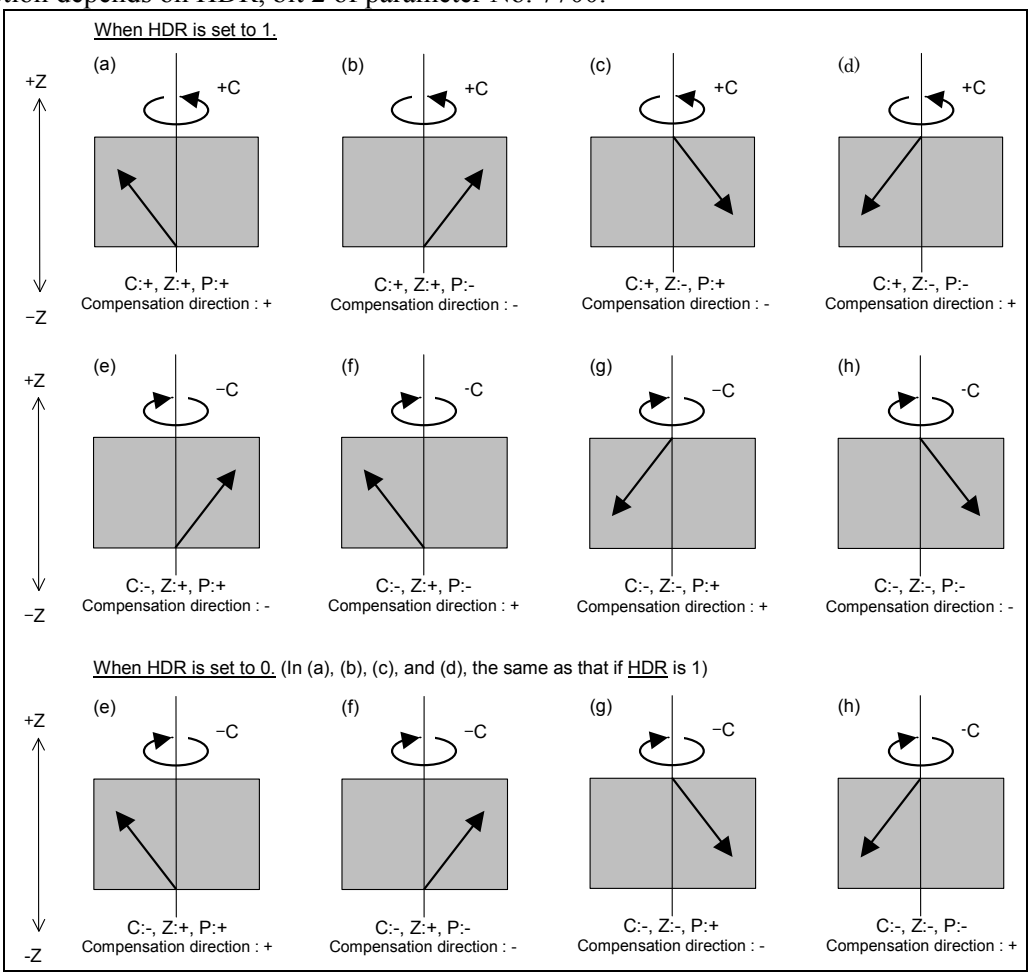


Fig. 21.6.2 (b) Direction of helical gear compensation

- Synchronization coefficient

A synchronization coefficient is internally represented using a fraction (K2/K1) to eliminate an error. The formula below is used for calculation.

$$\text{Synchronization coefficient} = \frac{K_2}{K_1} = \frac{L}{T} \times \frac{\beta}{\alpha}$$

where
L : Number of hob threads
T : Number of teeth
α : Number of pulses of the position detector per rotation about the master axis (parameter No. 7772)
β : Number of pulses of the position detector per rotation about the slave axis (parameter No. 7773)

K2/K1 is a value resulting from reducing the right side of the above formula, but the result of reduction is subject to the following restrictions:
$-32767 \leq K_2 \leq 32767$
$1 \leq K_1 \leq 32767$
When this restriction is not satisfied, the alarm (PS1596) is issued when G81 is specified.

The values of K_2 and K_1 are set to the parameter No. 4387 and No. 4388 automatically when G81 is specified. If T is not specified in G81 block, the alarm (PS1594) is issued. If L is not specified in G81 block, the synchronous ratio is calculated as L = 1.

Example)
When the pulses for one rotation (360000) are specified to the tool axis (master axis) on the following conditions, the pulses for position control are distributed as Fig. 21.6.2 (c).

Hob threads L : 10
Number of teeth of the work T : 100
Number of pulse per one revolution of position detector on the tool axis : 360000
Number of pulse per one revolution of position detector on the work axis : 360000

$$\text{Synchronization coefficient} = \frac{K_2}{K_1} = \frac{L}{T} \times \frac{\beta}{\alpha} = \frac{10}{100} \times \frac{360000}{360000} = \frac{1}{10}$$

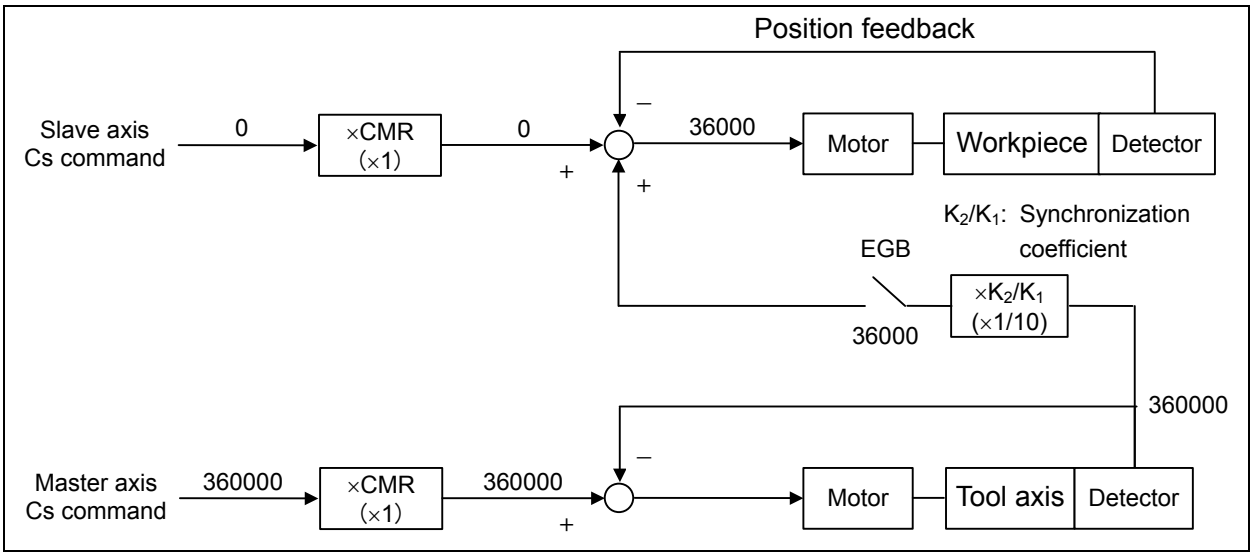


Fig. 21.6.2 (c) Command pulse distribution

As Fig. 21.6.2 (c), when 360000 pulses (Number of pulse for one rotation of the master axis) are specified, the pulses for slave axis by EGB are equal to the value which is multiplied to the number of pulse for one rotation of the slave axis by the ratio of Hob thread and number of teeth (rotation ratio between master axis and slave axis).
360000 × 1/10 = 36000

- Retract function

See Item, “Retract function” in the Subsection 21.6.1, “Electronic Gear Box”.

21.6.3 Electronic Gear Box Automatic Phase Synchronization

Overview

In the electronic gear box (EGB), when synchronization start or cancellation is specified, the synchronizing state is changed to another state gradually by applying acceleration/deceleration. This is because if synchronization is started or canceled immediately, this sudden speed change shocks a machine. Therefore, synchronization can be started or canceled while the spindle is rotating. Also, synchronization ratio can be changed while the spindle is rotating.

At the start of synchronization, automatic phase synchronization is performed such that the position where the machine coordinate is 0 about the workpiece axis matches the position corresponding to the spindle one-rotation signal. With this synchronization, the same operation is performed as synchronization start caused by a one-rotation signal in hobbing synchronization when using the functions of a hobbing machine.

The spindle corresponds to the EGB master axis and the workpiece axis corresponds to an EGB slave axis (servo axis or Cs contouring axis).

Format

M

- Acceleration/deceleration type

```
G81  T_  L_  R1  ;   Synchronization start
G80  R1  ;          Synchronization cancellation

 T :  Number of teeth (range of valid settings: 1-5000)
 L :  Number of hob threads (range of valid settings: -250 to +250, excluding 0)
      When L is positive, the direction of rotation about the workpiece axis is positive (+
      direction).
      When L is negative, the direction of rotation about the workpiece axis is negative (-
      direction).
```

- Acceleration/deceleration plus automatic phase synchronization type

```
G81  T_  L_  R2  ;   Synchronization start
G80  R1  ;          Synchronization cancellation

 T :  Number of teeth (range of valid settings: 1-5000)
 L :  Number of hob threads (range of valid settings: -250 to +250, excluding 0)
      When L is positive, the direction of rotation about the workpiece axis is positive (+
      direction).
      When L is negative, the direction of rotation about the workpiece axis is negative (-
      direction).
```

T

Set the following parameter when using in lathes.

Automatic phase synchronization is enabled in a command compatible with that for a hobbing machine used in lathes.

Bit 0 (EFX) of parameter No. 7731=1

Bit 5 (HBR) of parameter No. 7731=1

Bit 6 (PHS) of parameter No. 7702=1

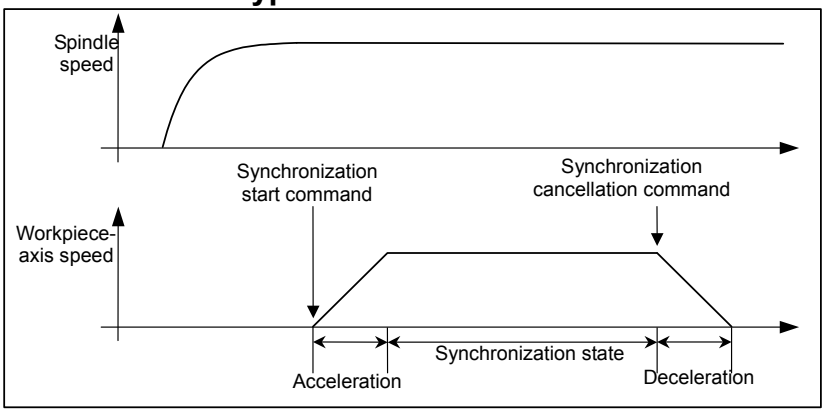
- **Acceleration/deceleration plus automatic phase synchronization type**

G81.4 R _ L _ ; Synchronization start
G80.4 ; Synchronization cancellation

R : Number of teeth (range of valid settings: 1-5000)
L : Number of hob threads (range of valid settings: -250 to +250, excluding 0)
When L is positive, the direction of rotation about the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation about the workpiece axis is negative (- direction).

Explanation

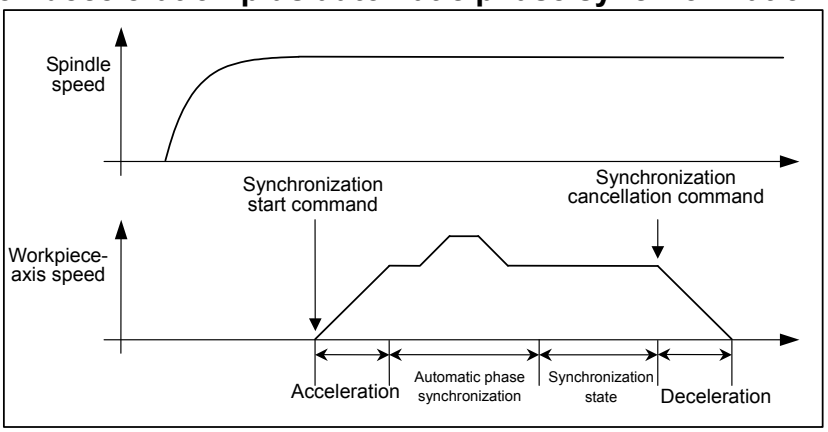
- **Acceleration/deceleration type**



1. Specify G81R1 to start synchronization.
   When G81R1 is specified, the workpiece axis (slave axis) is accelerated with the acceleration according to the acceleration rate set in the parameter No. 7778. When the speed reaches the synchronization speed, the G81R1 block is terminated.
2. For cancellation, specify G80R1 while the tool is moved away from the workpiece.
3. When G80R1 is specified, deceleration is started immediately at the acceleration rate set in parameter No. 7778.
   When the speed is reduced to 0, the G80R1 block is terminated.

NOTE
1 During synchronization start/cancellation, acceleration/deceleration is linear type.
2 In the automatic cancellation of synchronization due to one of the following causes, deceleration is performed and synchronization is canceled:
<1> Reset
<2> Alarm PW0000, “POWER MUST BE OFF”
<3> IO alarm
3 If bit 0 (EFX) of parameter No. 7731 is 0, the canned cycle for drilling cannot be used. To use the canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.

- Acceleration/deceleration plus automatic phase synchronization type



1. Move the workpiece axis to the position that corresponds to that of the one-rotation signal of the spindle.
2. Specify G81R2 to start synchronization.
   When G81R2 is specified, the workpiece axis is accelerated with the acceleration according to the acceleration rate set in the parameter No. 7778. Upon completion of phase synchronization, the G81R2 block terminates.
3. For cancellation, specify G80R2 while the tool is moved away from the workpiece.
4. When G80R2 is specified, deceleration is started immediately at the acceleration rate set in parameter No. 7778. When the speed is reduced to 0, the G80R2 block is terminated.

⚠ CAUTION

1 In automatic phase synchronization, specify the speed in parameter No. 7776 and the movement direction in parameter PHD, bit 7 of No. 7702.
2 In phase synchronization, rapid–traverse linear acceleration/deceleration (with the time constant specified in parameter No. 1620) is performed.
3 The workpiece–axis speed is obtained by superimposing the speed in automatic phase synchronization onto the speed corresponding to spindle rotation. In consideration of this superimposition, specify a position deviation limit in parameter No. 1828.

NOTE

1 The one-rotation signal used for automatic phase synchronization is issued not by the spindle position coder but by the separate pulse coder attached to the spindle and used to collect EGB feedback information. This means that the orientation position based on the one-rotation signal issued by the spindle position coder does not match the position used as the reference for the workpiece axis when phase synchronization is performed with automatic phase synchronization based on G81R2.
  Moreover, the one-rotation signal of the separate pulse coder must be turned on for each rotation of the spindle.
2 With the use of parameter No. 7777, the position at which the phase of the workpiece axis is matched can be shifted from the position corresponding to the one-rotation signal in automatic phase matching.
3 By setting bit 6 (EPA) of parameter No. 7731 to 1, automatic phase synchronization can be performed so that the workpiece axis position at the start of synchronization matches the position corresponding to the spindle one-rotation signal.

NOTE

4 By setting bit 6 (EPA) of parameter No. 7731 to 1, in automatic phase synchronization, when a synchronization command is issued again in the synchronization state, movement about the workpiece axis is made such that the position corresponding to the one-rotation signal of the spindle matches the position about the workpiece axis specified in the G81R2 synchronization start command executed first.
5 In automatic phase synchronization, movement is performed about the workpiece axis from the current position to the nearest phase position in the phase synchronization movement direction specified by the parameter.
6 Linear acceleration/deceleration applies to synchronization start/cancellation.
7 The acceleration/deceleration plus automatic phase synchronization type can be executed by the bit 6 (PHS) of parameter No. 7702, without specifying an R2 command in a G81 or G80 block.
8 In the automatic cancellation of synchronization due to one of the following causes, deceleration is performed and synchronization canceled:
<1> Reset
<2> Alarm PW0000, “POWER MUST BE OFF”
<3> IO alarm
9 When the spindle EGB is used and the control mode of the master axis is the speed control, it is necessary to execute the position coder orientation before that the automatic phase synchronization is started. In this case, set bit 7 (RFCHK3) of parameter No. 4016 for the master axis must be set ”0” to keep memorizing the one–rotation signal position of the spindle in the speed control mode.
10 When the spindle EGB is used and the control mode of the master axis is in the Cs contouring, it is necessary to execute the reference point return before the automatic phase synchronization is started.
11 The acceleration rate parameter No. 7778 must not be changed in the synchronization mode.
12 If parameter No. 7778 is 0, alarm PS1598 is issued when G81 is issued.
13 For Series 16*i*, the speed and the time constant for automatic phase synchronization are set separately in parameters Nos. 2135 and 2136(Nos. 4384 and 4385 when spindle EGB is used); in Series 30*i*, acceleration is directly set in parameter No. 7778.
14 If bit 0 (EFX) of parameter No. 7731 is 0, the canned cycle for drilling cannot be used. To use the canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.

Program example

- Acceleration/deceleration type

```
M03 ;                Clockwise spindle rotation command
G81  T_  L_  R1 ;    Start of synchronization
G00  X_ ;            Positions the workpiece at the machining position.

Machining in the synchronous state

G00  X_ ;            Retract the workpiece from the tool.
G81  T_  L_  R1 ;    Synchronization ratio change.
G00  X_ ;            Positions the workpiece at the machining position.
```

Machining in the synchronous state

```
G00  X_ ;          Retract the workpiece from the tool.
G80  R1  ;         Cancel synchronization
```

- Acceleration/deceleration plus automatic phase synchronization type

```
M03  ;  Clockwise spindle rotation command
G00  G90  C_ ;     C-axis positioning
G81  T_  L_  R2 ;  Start of synchronization.
G00  X_ ;          Positions the workpiece at the machining position.
```

Machining in the synchronous state

```
G00  X_ ;          Retract the workpiece from the tool.
G81  T_  L_  R2 ;  Synchronization ratio change.
G00  X_ ;          Positions the workpiece at the machining position.
```

Machining in the synchronous state

```
G00  X_ ;          Retract the workpiece from the tool.
G80  R2  ;         Cancel synchronization
```

21.6.4 Skip Function for EGB Axis

Overview

This function enables the skip or high-speed skip signal (these signals are collectively called skip signals in the remainder of this manual) for the EBG slave axis in synchronization mode with the EGB (electronic gear box).
This function has features such as the following:

1 If a skip signal is input while an EGB axis skip command block is being executed, this block does not terminate until the specified number of skip signals have been input.
2 If a skip signal is input while an EGB axis skip command block is being executed, the tool remains in synchronous mode and the EGB slave axis is not stopped and keeps moving.
3 The machine coordinates recorded when skip signals are input and the number of input skip signals are stored in specified custom macro variables.

Format

G81 T_ L_ ; **EGB mode ON**

G31.8 G91 α0 P_ Q_ (R_) ; **EGB skip command**

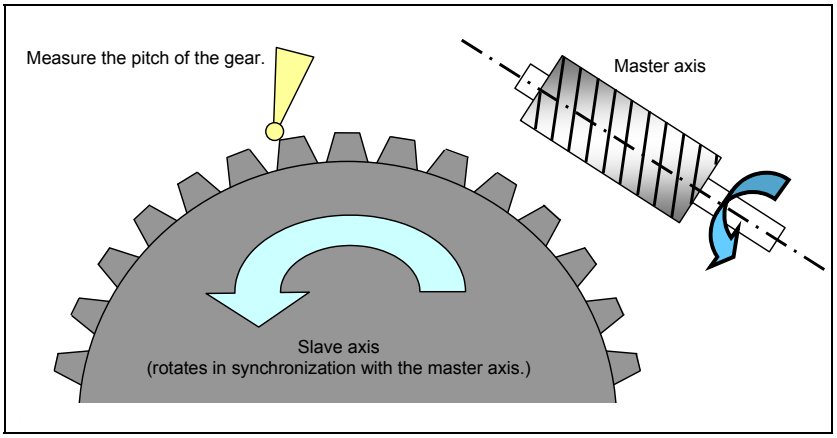
α: Specify an EGB slave axis. The specified value must always be "0".
P: Number of the first one of the custom macro variables used to store the machine coordinates recorded when skip signals are input.
Q: Number of skip signals that can be input during the execution of G31.8 (permissible range: 1 to 200).
R: Number of the custom macro variable used to store the number of input skip signals.
Specify it to check the number of input signals.

Explanation

G31.8 is a one-shot G code.
After the execution of G31.8, values of machine coordinate which is gotten at each time of skip signal input are set in custom macro variables. The numbers of variables are used from the top number commanded by P to the number added with the amount of times commanded by Q.
And the total times of skip signal input is set in the custom macro variable whose number is commanded by R.

Example

The pitch of a gear can be measured.




```
G81 T200 L2 ; .....EGB mode ON
X_ ;
Z_ ;
G31.8 G91 C0 P500 Q200 R1 ;.....EGB skip command
```

After 200 skip signals have been input, the 200 skip positions on the C-axis that correspond to the respective skip signals are stored in custom macro variables #500 to #699.
Also, the number of input skip signals is stored in custom macro variable #1.

NOTE

1 When specifying this function, specify only a single EGB slave axis. If no axis is specified for two or more axes are specified, alarm PS1152 is generated.
2 If P is not specified, alarm PS1152 is generated.
3 If R is not specified, the number of input skip signals is not written to a custom macro variable.
4 The custom macro variable numbers specified in P and R must be existing ones. If a non-existent variable number is specified, alarm PS0115 is generated. If a variable shortage occurs, alarm PS0115 is generated.
5 Whether to use conventional skip signals or high-speed skip signals with this function can be specified with HSS, bit 4 of parameter No. 6200. If high-speed skip signals are used, specify which high-speed signals to enable with 9S1 to 9S8, bits 0 to 7 of parameter No. 6208.
6 Skip positions are calculated from feedback pulses from the machine. Thus, they are free from errors due to delay in acceleration/deceleration and the servo system.

21.6.5 Electronic Gear Box 2 Pair

Overview

This function enables machining of high-precision gears, screws, and other components by rotating the workpiece in synchronization with a rotating tool or by moving the tool in synchronization with a rotating workpiece. The rate of synchronization can be specified with a program.
Up to two sets of axes can be synchronized. A gear grinding machine can be controlled, for instance, by using one axis for rotating the workpiece in synchronization with the tool and another axis for performing dressing in synchronization with the tool.
The specification method differs depending on the configuration of the machine. For details, refer to the relevant manual provided by the machine tool builder.
In the subsequent explanation, the Electronic Gear Box is called the EGB.

21.6.5.1 Specification method (G80.5, G81.5)

Format

G81.5 $\left\{\begin{matrix}\mathbf{T}\,t\\ \mathbf{P}\,p\end{matrix}\right\}$ $\left\{\begin{matrix}\beta\,j\\ \beta\,\mathbf{0}\ \mathbf{L}\,l\end{matrix}\right\}$ **;** Start of synchronization

↑ ↑

Amount of travel along the master axis

Amount of travel along the slave axis

G80.5 β0 ; Cancellation of synchronization

Explanation

- Master axis, slave axis, and dummy axis

The synchronization reference axis is called the master axis, while the axis along which movement is performed in synchronization with the master axis is called the slave axis. For example, if the workpiece moves in synchronization with the rotating tool as in a hobbing machine, the tool axis is the master axis and the workpiece axis is the slave axis.
Which axes to become the master and slave axes depends on the configuration of the machine. For details, refer to the manual issued by the machine tool builder.
A single servo axis is used exclusively so that digital servo can directly read the rotation position of the master axis. (This axis is called the EGB dummy axis.)

- Start of synchronization

When the ratio of the master-axis travel to the slave-axis travel is specified, synchronization starts.

Specify the master-axis travel in either of the following ways.

1 Master-axis speed
T t: Master-axis speed (1≤ t ≤5000)

2 Master-axis pulse count
P p : Master-axis pulse count (1≤ p ≤999999999)
For an A/B phase detector, specify the count as four pulses corresponding with one A/B phase cycle.

Specify the slave-axis travel in either of the following ways.

1 Slave-axis travel
βj : Slave-axis address
j : Slave-axis travel indicated in units of the least command increment (the maximum specifiable range is equal to usual axis)

When j = 0, the specified command is regarded as being a command for the slave-axis speed, described below. In this case, if L is not specified, an alarm is output.

2 Slave-axis speed
β0 L±*l*
β : Slave-axis address
l : Slave axis speed(-250≤ *l* ≤250, but *l* = 0 is excluded.)

- Cancellation of synchronization

1 Canceling synchronization for each axis by issuing a command With a G80.5 β0 command, synchronization is canceled.
β is the address of the slave axis. Synchronization of the slave axis specified by β is canceled. A cancellation command can be issued only for one axis in one block.
When β0 is not specified, the synchronization of all currently synchronized axes is canceled.
When cancellation of synchronization is issued, the absolute coordinate on the workpiece axis is updated in accordance with the amount of travel during synchronization. For a rotation axis, the amount of travel during synchronization is rounded within 360-degree and added to the absolute coordinate.

2 Canceling synchronization by a reset
By setting HBR, bit 0 of parameter No. 7700, to 0, synchronization is canceled by a reset.

3 Others
Synchronization is automatically canceled under the following conditions.
<1> An emergency stop is applied.
<2> A servo alarm is generated.
<3> Alarm PW0000 (POWER MUST BE OFF) is generated.
<4> An IO alarm is generated.

⚠ CAUTION
1 Feed hold, interlock, and machine lock are invalid to a slave axis in EGB synchronization.
2 Even if an OT alarm is issued for a slave axis in EGB synchronization, synchronization will not be canceled.
3 During synchronization, it is possible to execute a move command for a slave axis and other axes, using a program. The move command for a slave command must be an incremental one.

NOTE
1 If bit 0 (HBR) of parameter No. 7700 is set to 1, EGB synchronization will not be canceled due to a reset. Usually, set this parameter bit to 1.
2 In synchronous mode, it is not possible to specify G27, G28, G29, G30, G30.1, and G53 for a slave axis.
3 It is not possible to use controlled axis detach for a slave axis.
4 During synchronization, manual handle interruption can be performed on the slave and other axes.
5 In synchronization mode, no inch/metric conversion commands (G20 and G21) can be issued.
6 In synchronous mode, only the machine coordinates on a slave axis are updated.

NOTE

7 If bit 0 (EFX) of parameter No. 7731 is 0, no canned cycle for drilling can be used. To use a canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1.

8 If, during synchronization, G81.5 is issued again, alarm PS1595 is issued if bit 3 (ECN) of parameter No. 7731 is 0. If bit 3 (ECN) of parameter No. 7731 is 1, the synchronization coefficient can be changed to a newly specified one.

9 Actual cutting feedrate display does not take synchronization pulses into consideration.

10 For an EGB slave axis, synchronous and composite control cannot be executed.

11 In EGB synchronization mode, AI contour control mode is temporarily canceled.

21.6.5.2 Description of commands compatible with those for a hobbing machine (G80, G81)

A command compatible with that for a hobbing machine can be used as a synchronization command.
Usually, a hobbing machine performs machining by synchronizing the workpiece axis (usually, the C-axis) to the hobbing axis (spindle).
If there are two synchronization sets with the EGB, which synchronization set to start with this specification method can be specified with parameter No. 7710.

Format

	Parameter EFX(No.7731#0)=0	Parameter EFX(No.7731#0)=1	
		Parameter HBR(No.7731#5)=1	Parameter HBR(No.7731#5)=0
Start of synchronization	**G81 T__ (L__) (Q__ P__) ;**	**G81.4 R__ (L__) (Q__ P__) ;**	**G81.4 T__ (L__) (Q__ P__) ;**
Cancellation of synchronization	**G80 ;**	**G80.4 ;**	**G80.4 ;**
	(*1) (*4)	**(*2) (*4)**	**(*3) (*4)**

T(or R) : Number of teeth (Specifiable range: 1 to 5000)
L : Number of hob threads (Specifiable range: -250 to +250)
The sign of L determines the direction of rotation for the workpiece axis.
When L is positive, the direction of rotation for the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation for the workpiece axis is negative (- direction).
When L is 0, it depends on the setting of bit 3 (LZR) of parameter No. 7701.
If L is not specified, the number of hob threads is assumed to be1.
Q : Module or diametral pitch
Specify a module in the case of metric input.
(Unit: 0.00001mm, Specifiable range: 0.01 to 25.0mm)
Specify a diametral pitch in the case of inch input.
(Unit: 0.00001inch^{-1}, Specifiable range: 0.01 to 254.0 inch^{-1})
P : Gear helix angle
(Unit: 0.0001deg, Specifiable range: -90.0 to +90.0deg)
*1 Use it for machining centers.
*2 Use it for lathes.
*3 Use it for machining centers.
This format enables specification of the same G codes as those for lathes.
*4 When specifying Q and P, the user can use a decimal point.

NOTE
Specify G81, G80, G81.4, and G80.4 in a single block.

Explanation

- Start of synchronization

Specify P and Q to use helical gear compensation. In this case, if only one of P and Q is specified, alarm PS1594 is generated.
When G81 is issued so that the machine enters synchronization mode, the synchronization of the workpiece axis to the spindle is started.

During synchronization, the rotation about the tool and workpiece axes is controlled so that the relationship between T (number of teeth) and L (number of hob threads) is maintained.
If, during synchronization, G81 is issued again without synchronization cancellation, alarm PS1595 is generated if ECN, bit 3 of parameter No. 7731, is 0. If ECN, bit 3 of parameter No. 7731, is 1, helical gear compensation is performed with the synchronization coefficient being changed to the one newly specified with T and L commands if T and L commands are issued. If T and L commands are not issued and only P and Q commands are issued, helical gear compensation is performed with the synchronization coefficient kept intact. This allows consecutive machining of helical gears and spur gears.

- Cancellation of synchronization

Synchronization of all synchronized axes is canceled.
When cancellation of synchronization is issued, the absolute coordinate on the workpiece axis is updated in accordance with the amount of travel during synchronization.
For a rotation axis, the amount of travel during synchronization is rounded within 360-degree and added to the absolute coordinate.
In the G80 block, only O and N addresses can be specified.

⚠ CAUTION

1 Feed hold, interlock, and machine lock are invalid to a slave axis in EGB synchronization.
2 Even if an OT alarm is issued for a slave axis in EGB synchronization, synchronization will not be canceled.
3 During synchronization, it is possible to execute a move command for a slave axis and other axes, using a program. The move command for a slave command must be an incremental one.

NOTE

1 If bit 0 (HBR) of parameter No. 7700 is set to 1, EGB synchronization will not be canceled due to a reset. Usually, set this parameter bit to 1.
2 In synchronous mode, it is not possible to specify G27, G28, G29, G30, G30.1, and G53 for a slave axis.
3 It is not possible to use controlled axis detach for a slave axis
4 During synchronization, manual handle interruption can be performed on the slave and other axes.
5 In synchronization mode, no inch/metric conversion commands (G20 and G21) cannot be issued.
6 In synchronous mode, only the machine coordinates on a slave axis are updated.
7 If bit 0 (EFX) of parameter No. 7731 is 0, no canned cycle for drilling can be used. To use a canned cycle for drilling, set bit 0 (EFX) of parameter No. 7731 to 1 and use G81.4 instead of G81 and G80.4 instead of G80.
8 If bit 0 (TDP) of parameter No. 7702 is 1, the permissible range of T is 0.1 to 100 (1/10 of the specified value).
9 If, at the start of EGB synchronization (G81), L is specified as 0, synchronization starts with L assumed to be 1 if bit 3 (LZR) of parameter No. 7701 is 0; if bit 3 (LZR) of parameter No. 7701 is 1, synchronization is not started with L assumed to be 0. At this time, helical gear compensation is performed.
10 Feed per revolution is performed on the feedback pulses on the spindle. By setting bit 0 (ERV) of parameter No. 7703 to 1, feed per revolution can be performed based on the speed on the synchronous slave axis.

NOTE
11 Actual cutting feedrate display does not take synchronization pulses into consideration.
12 For an EGB slave axis, synchronous and composite control cannot be executed.
13 In EGB synchronization mode, AI contour control mode is temporarily canceled.

- Helical gear compensation

For a helical gear, the workpiece axis is compensated for the movement along the Z axis (axial feed axis) based on the torsion angle of the gear.
Helical gear compensation is performed with the following formulas:

$$\text{Compensation angle} = \frac{Z \times \sin(P)}{\pi \times T \times Q} \times 360 \text{ (for metric input)}$$

$$\text{Compensation angle} = \frac{Z \times Q \times \sin(P)}{\pi \times T} \times 360 \text{ (for inch input)}$$

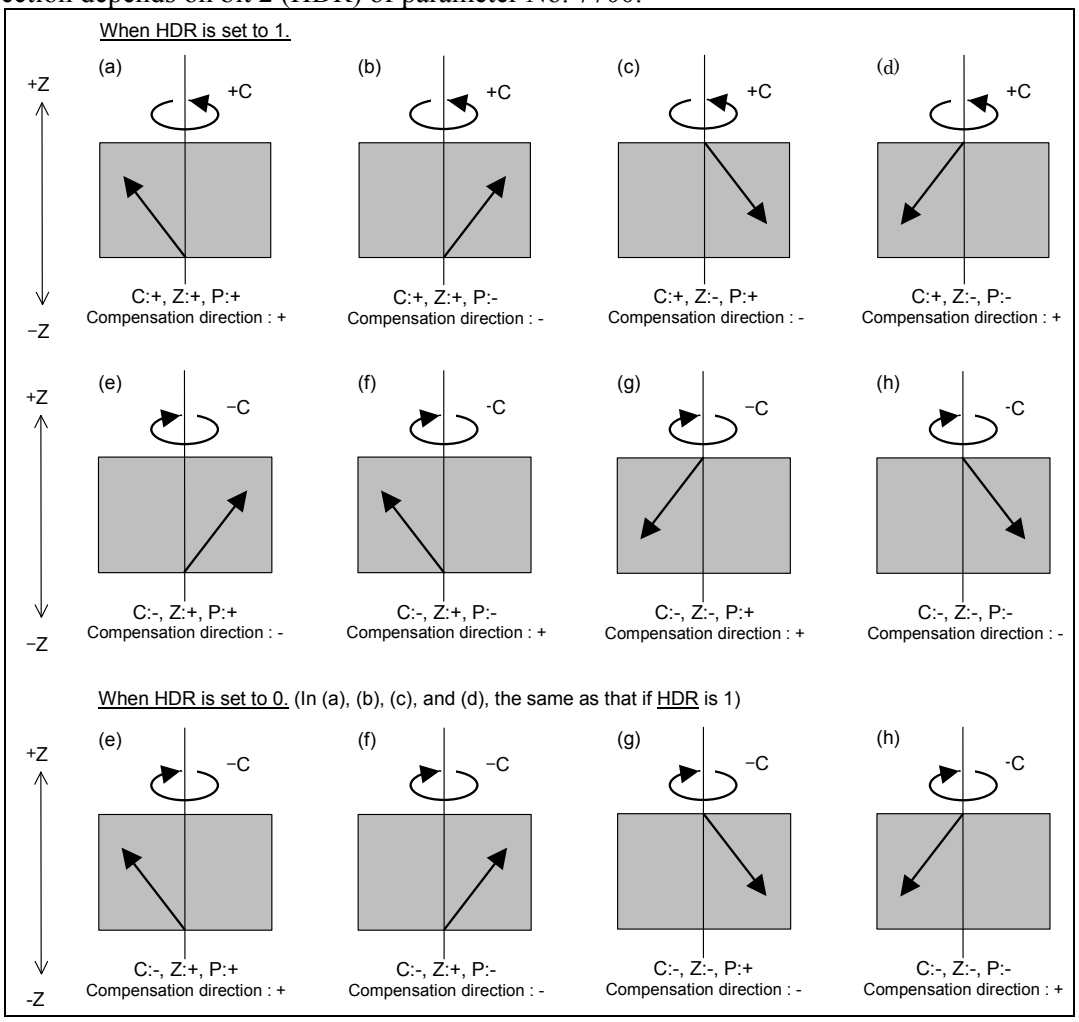
where

Compensation angle: Signed absolute value (deg)
Z : Amount of travel on the Z-axis after the specification of G81 (mm or inch)
P : Signed gear helix angle (deg)
π : Circular constant
T : Number of teeth
Q : Module (mm) or diametral pitch ($inch^{-1}$)
Use P, T, and Q specified in the G81 block.

In helical gear compensation, the machine coordinates on the workpiece axis and the absolute coordinates are updated with helical gear compensation.

- Direction of helical gear compensation

The direction depends on bit 2 (HDR) of parameter No. 7700.



21.6.5.3 Controlled axis configuration example

- For gear grinders

Spindle : EGB master axis : Tool axis
1st axis : X axis
2nd axis : Y axis
3rd axis : C axis (EGB slave axis : Workpiece axis)
4th axis : C axis (EGB dummy axis : Cannot be used as a normal controlled axis)
5th axis : V axis (EGB slave axis : Dressing axis)
6th axis : V axis (EGB dummy axis : Cannot be used as a normal controlled axis)

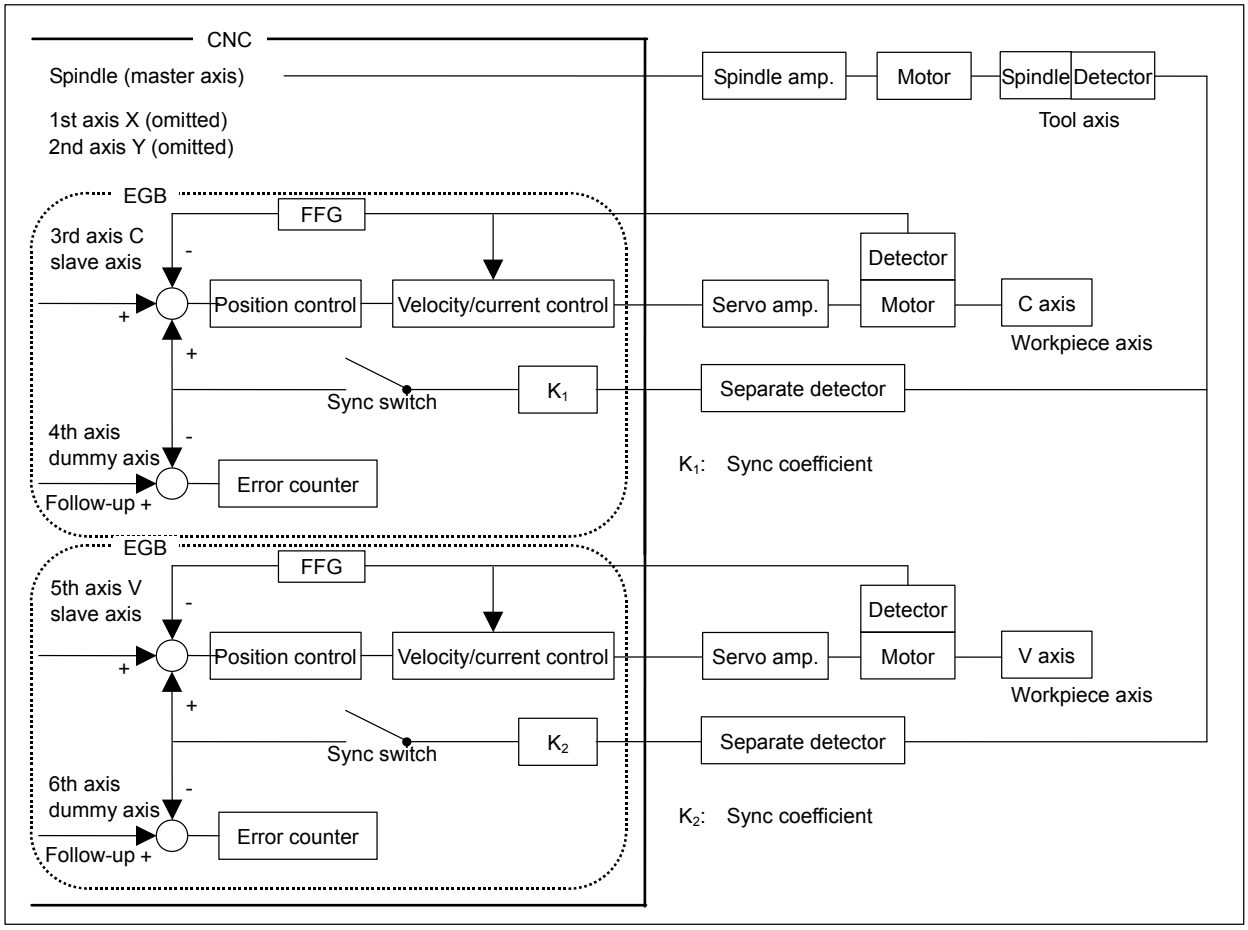


Fig. 21.6.5.3 (a) Controlled axis configuration with a set of two EGBs

21.6.5.4 Sample programs

- When the master axis is the spindle, and the slave axis is the C-axis

(1) G81.5 T10 C0 L1 ;
Synchronization between the master axis and C-axis is started at the ratio of one rotation about the C-axis to ten rotations about the master axis.

(2) G81.5 T10 C0 L-1 ;
Synchronization between the master axis and C-axis is started at the ratio of one rotation about the C-axis to ten rotations about the master axis.
In this case, however, the direction of rotation is opposite to that of (1).

(3) G81.5 T1 C3.26 ;
Synchronization between the master axis and C-axis is started at the ratio of a 3.26-degree rotation about the C-axis per one rotation about the master axis.

(4) G81.5 P10000 C-0.214 ;
Synchronization between the master axis and C-axis is started at the ratio of a -0.214 degree rotation about the C-axis to 10,000 feedback pulses from the pulse coder of the master axis.

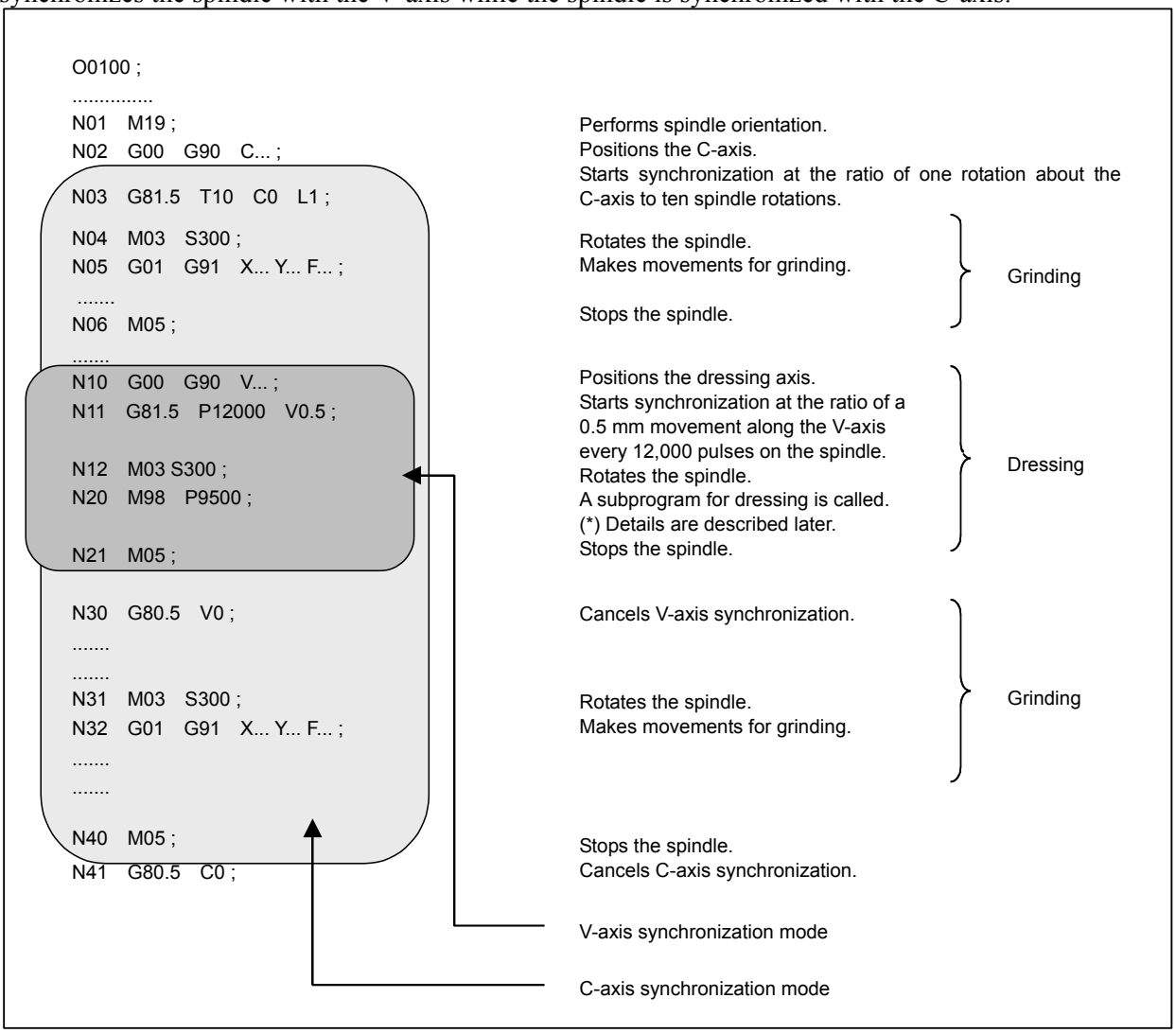
- When the master axis is the spindle, the slave axis is the V-axis (linear axis), and inch/metric conversion is performed

(1) For a millimeter machine and metric input
G81.5 T1 V1.0 ;
Synchronization between the master axis and V-axis is started at the ratio of a 1.00 mm movement along the V-axis per rotation about the master axis.

(2) For a millimeter machine and inch input
G81.5 T1 V1.0 ;
Synchronization between the master axis and V-axis is started at the ratio of a 1.0 inch movement (25.4 mm) along the V-axis per rotation about the master axis.

- When two groups of axes are synchronized simultaneously

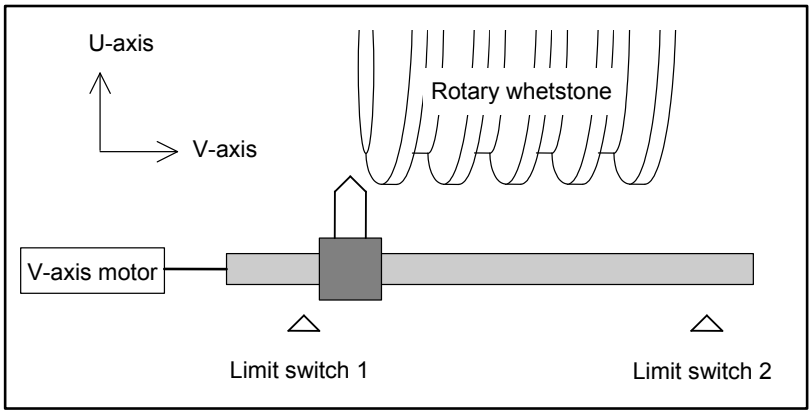
Based on the controlled axis configuration described in Fig. 21.6.5.3 (a), the sample program below synchronizes the spindle with the V-axis while the spindle is synchronized with the C-axis.



Thus, the synchronizations of each group can be started and canceled independently.

- Example of use of dressing

Gear grinder in the following machine configuration




```
O9500 ;
N01  G01  G91  U_  F100 ;     Dressing axis approach
N02  M03  S100 ;               The M03 command causes the PMC to rotate the whetstone in the
                               positive direction.
                               In accordance with this, the tool moves along the V-axis in the
                               positive direction. When the tool reaches the position of limit switch
                               2 on the V-axis, the PMC stops the whetstone and returns FIN.
N02  U_  V_ ;                  Movement to the next dressing position
N03  M04  S100 ;               The M04 command causes the PMC to rotate the whetstone in the
                               negative direction.
                               In accordance with this, the tool moves along the V-axis in the
                               negative direction. When the tool reaches the position of limit switch
                               1 on the V-axis, the PMC stops the whetstone and returns FIN.
N04  U_ V_ ;                   Movement to the next dressing position
                               If required, N02 to N04 are repeated to conduct dressing.
.....
.....

M99 ;
```

NOTE

If the V-axis (linear axis) is synchronized with the spindle as in dressing, the V-axis travel range is determined by the rotation of the spindle. To perform dressing with the tool moving back and forth along the V-axis in a certain range, therefore, the PMC must perform an operation in which the tool is stopped temporarily and is reversed when it reaches a certain position on the V-axis.
In the above example, limit switches are provided to determine the range of travel along the V-axis and the PMC performs control so that the whetstone rotates until the tool reaches the position of each limit switch on the V-axis.
By using the position switch function instead of limit switches, dressing can be performed as in the following example, without the need to mount limit switches to the machine. By rewriting the operating ranges of the position switches (parameters Nos. 6930 to 6945 and Nos. 6950 to 6965) using the G10 programmable parameter input, the range of travel along the V-axis can be specified using a program.

- Command specification for hobbing machines

Based on the controlled axis configuration described in Fig. 21.6.5.3 (a), the sample program below sets the C-axis (in parameter 7710) for starting synchronization with the spindle according to the command specification method for hobbing machines.

```
O1234 ;
N0010  M19  ;              Tool axis orientation
N0020  G28  G91  C0  ;     Reference position return on the work axis
N0030  G81  T20  L1 ;      Start of synchronization of the tool axis and the C-axis
                           (Rotation about the C-axis by 18° per rotation about the tool axis)
N0040  S300  M03 ;         Rotation about the tool axis at 300 min-1.
N0050  G01  X_  F_ ;       Movement along the X-axis (cutting).
N0060  G01  Y_  F_ ;       Movement along the Y-axis (grinding).
     ;                     If required, C, X, Y, and other axis command are possible.
     ;
N0100  G01  X_  F_ ;       Movement along the X-axis (escape)
N0110  M05 ;               Stop on the tool axis.
N0120  G80 ;               Cancellation of synchronization of the tool and C-axes.
N0130  M30 ;
```

21.6.5.5 Synchronization ratio specification range

The programmed ratio (synchronization ratio) of a movement along the slave axis to a movement along the master axis is converted to a detection unit ratio inside the NC automatically. If such converted data (detection unit ratio) exceeds a certain allowable data range in the NC, synchronization cannot be maintained correctly, and an alarm PS1596 is issued.
Even when a programmed movement of master axis and a slave axis is within specifiable ranges, a detection unit ratio obtained by conversion can exceed the allowable range. This causes an alarm.
Let K be a synchronization ratio. The internal data corresponding K is the amount of slave axis movement (Kn) represented in the detection unit divided by the amount of master axis movement (Kd) represented in the detection unit;
this fraction is represented as Kn/Kd (reduced to its lowest terms) as indicated below.

$$K = \frac{Kn}{Kd} = \frac{\text{Amount of slave axis movement represented in the detection unit}}{\text{Amount of master axis movement represented in the detection unit}}$$

Kn and Kd must lie within the following ranges:

$$-2147483648 \le Kn \le 2147483647$$
$$1 \le Kd \le 2147483647$$

When Kn or Kd exceeds its allowable range above, an alarm PS1596 is issued.
In conversion to the detection unit, when the CMR (command multiplication: parameter 1820) is a fraction or when inch/millimeter conversion is used, the fraction is directly converted without modification so that no error can occur in the conversion of specified amounts of movement.
During conversion, the amount of movement is multiplied by 254/100 for inch input on a millimeter machine, and 100/254 for metric input on an inch machine. Thus, Kn and Kd can become large numbers. If a synchronization ratio cannot be reduced to its lowest terms, an alarm condition is likely to occur.

- Example 1)

Based on the controlled axis configuration described in Fig. 21.6.5.3 (a), suppose that the spindle and V-axis are as follows:
Spindle pulse coder : 72000pulse/rev (4 pulses for one A/B phase cycle)
C-axis least command increment : 0.001deg
C-axis CMR : 5

V-axis least command increment : 0.001mm
V-axis CMR : 5

Then, the C-axis detection unit is 0.0002 degree. The V-axis detection unit is 0.0002 mm. In this case, the synchronization ratio (Kn, Kd) is related with a command as indicated below. Here, let Pm and Ps be the amounts of movements represented in the detection unit on the master axis and slave axis specified in a synchronization start command, respectively.

(1) When the master axis is the spindle, and the slave axis is the C-axis

(a) Command : G81.5 T10 C0 L1 ;
Operation : Synchronization between the spindle and C-axis is started at the ratio of one rotation about the C-axis to ten spindle rotations.
Pm : (Number of pulses per spindle rotation) × (10 rotations) → 72000 × 10
Ps : (Amount of movement per rotation about the C-axis) × CMR × (one rotation) → 360000 × 5 × 1

$$\frac{Kn}{Kd} = \frac{360000 \times 5 \times 1}{72000 \times 10} = \frac{5}{2}$$

Both Kn and Kd are within the allowable range. No alarm is output.

(b) Command : G81.5 T10 C0 L-1 ;
Operation : Synchronization between the spindle and C-axis is started at the ratio of one rotation about the C-axis to ten spindle rotations.
In this case, however, the direction of rotation is opposite to that of (a) above.
Pm : (Number of pulses per spindle rotation) × (10 rotations) → 72000 × 10
Ps : (Amount of movement per rotation about the C-axis) × CMR × (one rotation) → -360000 × 5 × 1

$$\frac{Kn}{Kd} = \frac{-360000 \times 5 \times 1}{72000 \times 10} = \frac{-5}{2}$$

Both Kn and Kd are within the allowable range. No alarm is output.

(c) Command : G81.5 T1 C3.263 ;
Operation : Synchronization between the spindle and C-axis is started at the ratio of a 3.263-degree rotation about the C-axis to one spindle rotation.
Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1
Ps : (Amount of C-axis movement) × CMR → 3263 × 5

$$\frac{Kn}{Kd} = \frac{3263 \times 5}{72000 \times 1} = \frac{3263}{14400}$$

Both Kn and Kd are within the allowable range. No alarm is output.

In this sample program, when T1 is specified for the master axis, the synchronization ratio (fraction) of the CMR of the C-axis to the denominator Kd can always be reduced to lowest terms, thus Kd falls in the allowable range. So, the specifiable range of C is as follows:

$-99999999 \leq C \leq 99999999$

(d) Command : G81.5 T10 C3.263 ;
Operation : Synchronization between the spindle and C-axis is started at the ratio of a 3.263-degree rotation about the C-axis to ten spindle rotations.
Pm : (Number of pulses per spindle rotation) × (10 rotations) → 72000 × 10
Ps : (Amount of C-axis movement) × CMR → 3263 × 5

$$\frac{Kn}{Kd} = \frac{3263 \times 5}{72000 \times 10} = \frac{3263}{144000}$$

In this case, an alarm is issued because Kd exceeds the specifiable range.

(e) Command : G81.5 P10000 C-0.214 ;
Operation : Synchronization between the spindle and C-axis is started at the ratio of a -0.214 degree rotation of the C-axis to 10,000 feedback pulses from the pulse coder of the spindle.
Pm : (Specified number of feedback pulses from the pulse coder of the spindle) → 10000
Ps : (Amount of C-axis movement) × CMR → -214 × 5

$$\frac{Kn}{Kd} = \frac{-214 \times 5}{10000} = \frac{-107}{1000}$$

Both Kn and Kd are within the allowable range. No alarm is output.

(2) When the master axis is the spindle, the slave axis is the V-axis (linear axis), and inch/metric conversion is performed

(a) For a millimeter machine and metric input
Command : G81.5 T1 V1.0 ;
Operation : Synchronization between the spindle and V-axis is started at the ratio of a 1.0 mm movement along the V-axis per spindle rotation.
Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1
Ps : (Amount of V-axis movement) × CMR → 1000 × 5

$$\frac{Kn}{Kd} = \frac{1000 \times 5}{72000} = \frac{5}{72}$$

Both Kn and Kd are within the allowable range. No alarm is output.

(b) For a millimeter machine and inch input
Command : G81.5 T1 V1.0 ;
Operation : Synchronization between the spindle and V-axis is started at the ratio of a 1.0inch (25.4mm) movement along the V-axis per spindle rotation.
Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1
Ps : (Amount of V-axis movement) × CMR × 254 ÷ 100 → 10000 × 5 × 254 ÷ 100

$$\frac{Kn}{Kd} = \frac{10000 \times 5 \times 254}{72000 \times 100} = \frac{127}{72}$$

Both Kn and Kd are within the allowable range. No alarm is output.

(c) For a millimeter machine and inch input
Command : G81.5 T1 V0.0013 ;
Operation : Synchronization between the spindle and V-axis is started at the ratio of a 0.0013 inch (0.03302 mm) movement along the V-axis per spindle rotation.
Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1
Ps : (Amount of V-axis movement) × CMR × 254 ÷ 100 → 13 × 5 × 254 ÷ 100

$$\frac{Kn}{Kd} = \frac{13 \times 5 \times 254}{72000 \times 100} = \frac{1651}{720000}$$

In this case, an alarm is issued because Kd exceeds the specifiable range.

- Example 2)

Based on the controlled axis configuration described in Fig. 21.6.5.3 (a), suppose that the spindle and V-axis are as follows:

Spindle pulse coder : 72000pulse/rev (4 pulses for one A/B phase cycle)
C-axis least command increment : 0.001deg
C-axis CMR : 1/2
V-axis least command increment : 0.001mm
V-axis CMR : 1/2

Then, the C-axis detection unit is 0.002 degree. The V-axis detection unit is 0.002 mm.

In this case, the synchronization ratio (Kn, Kd) is related with a command as indicated below. Here, let Pm and Ps be the amounts of movements represented in the detection unit for the master axis and slave axis specified in a synchronization start command, respectively.

(1) When the master axis is the spindle, and the slave axis is the C-axis

(a) Command : G81.5 T1 C3.263 ;
Operation : Synchronization between the spindle and C-axis is started at the ratio of a 3.263-degree rotation about the C-axis per spindle rotation.
Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1
Ps : (Amount of C-axis movement) × CMR → 3263 × 1 ÷ 2

$$\frac{Kn}{Kd} = \frac{3263 \times 1}{72000 \times 2} = \frac{3263}{144000}$$

In this case, an alarm is issued because Kd exceeds the specifiable range.

(b) Command : G81.5 T1 C3.26 ;
The value specified for C slightly differs from that specified for C in (a).
Operation : Synchronization between the spindle and C-axis is started at the ratio of a 3.26-degree rotation about the C-axis per spindle rotation.
Pm : (Number of pulses per spindle rotation) × (one rotation) → 72000 × 1
Ps : (Amount of C-axis movement) × CMR → 3260 × 1 ÷ 2

$$\frac{Kn}{Kd} = \frac{3260 \times 1}{72000 \times 2} = \frac{163}{7200}$$

(a) causes an alarm to be output because the values cannot be reduced. (b) causes no alarm because the ratio of the travel distances can be reduced to a simple ratio.

21.6.5.6 Retract function

See "Retract function" in the Subsection 21.6.1, "Electronic Gear Box".

21.6.6 U-axis Control

Overview

Conventionally, the control of an axis on a spindle, such as the U-axis of a vertical lathe, from a motor mounted in a location other than the spindle has required a mechanism, consisting of a planetary gear box and differential gears, to prevent the axis from moving as the spindle is rotating.

The U-axis control function enables the U-axis to remain in a fixed position or to move at a programmed speed without using a mechanism such as a planetary gear box. This is done by causing the U-axis motor to rotate in such a way that U-axis movement, which would be caused by the rotation of the spindle, is canceled out. For details, refer to the manual supplied by the machine tool builder.

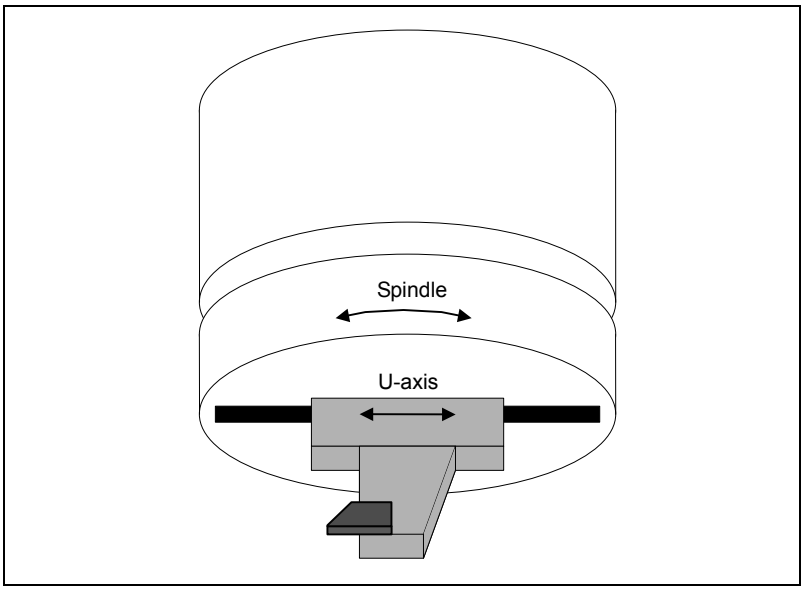


Fig. 21.6.6 (a) Example of a machine having the U-axis

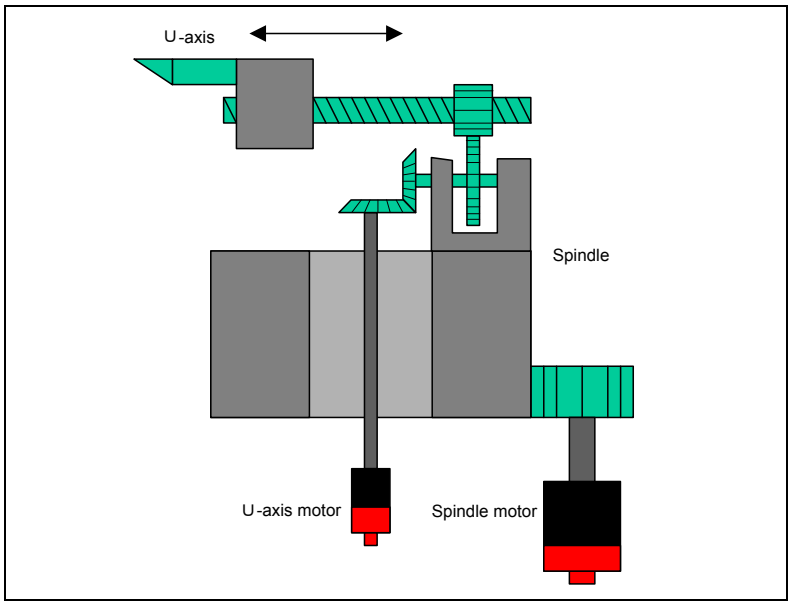


Fig. 21.6.6 (b) Example of the structure of a machine having the U-axis

In the example of the above structure, the tool moves along the U-axis when the spindle rotates. This movement is canceled out by rotating the U-axis motor.

21.6.7 U-axis Control 2 Pairs

Overview

The U-axis control 2 pairs function enables the U-axis to remain at a fixed position or to move at a programmed speed without using a mechanism such as a planetary gear box. The U-axis movement, which would be caused by the rotation of the spindle, is canceled out by rotating the U-axis motor.
Two synchronous pairs can be specified in this function.

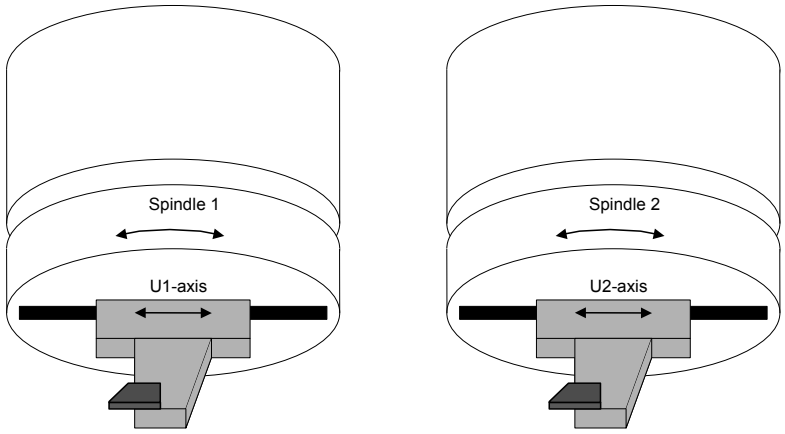


Fig. 21.6.7 (a) Example of a machine having two U-axis pairs

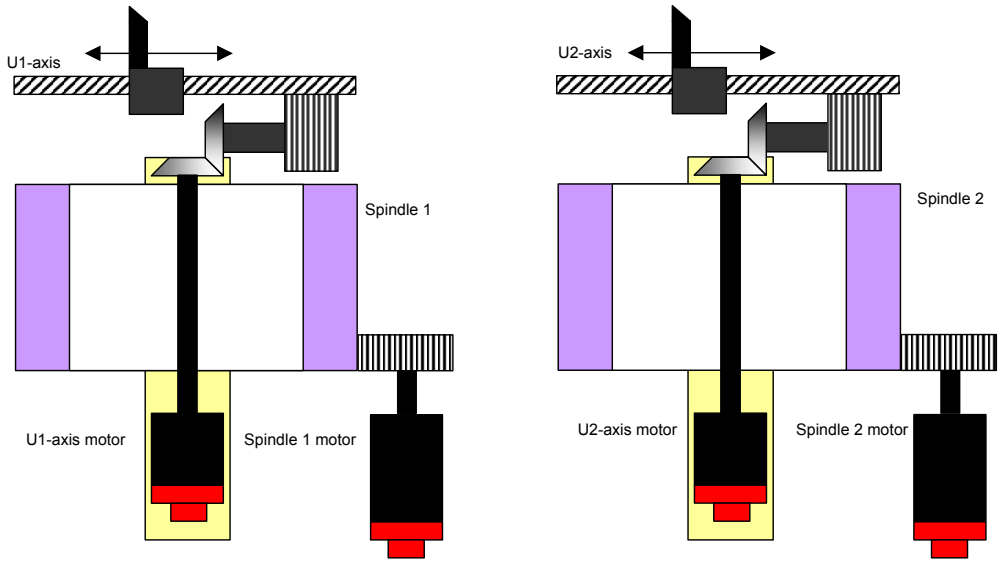


Fig. 21.6.7 (b) Example of the mechanism of a machine having two U-axis pairs

In the above mechanism example, when spindles 1 and 2 rotate, the U1 and U2 axes are synchronized with the spindles.

21.7 TANDEM CONTROL

When enough torque for driving a large table cannot be produced by only one motor, two motors can be used for movement along a single axis. Positioning is performed by the main motor only. The submotor is used only to produce torque. With this tandem control function, the torque produced can be doubled.

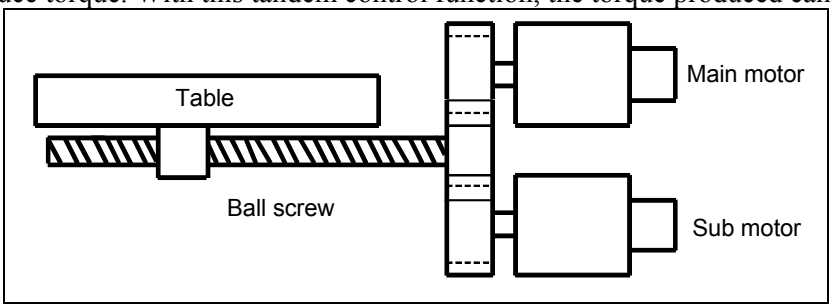


Fig. 21.7 (a) Example of operation

In general, the NC regards tandem control as being performed for one axis. However, for servo parameter management and servo alarm monitoring, tandem control is regarded as being performed for two axes.
For details, refer to the relevant manual published by the machine tool builder.

21.8 PIVOT AXIS CONTROL

Overview

This function is intended to control the pivot axis (B-axis), using the ball screw connected to the servo motor, which has a degree of freedom, as shown in Fig. 21.8 (a).

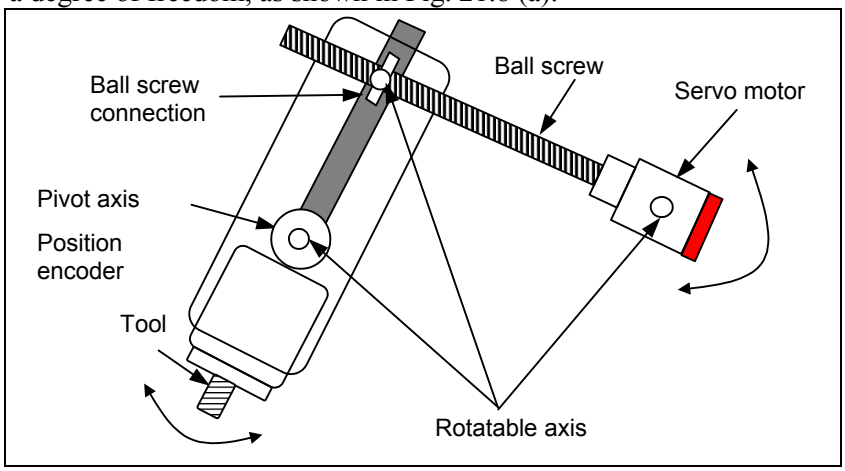


Fig. 21.8 (a)

For the rotation axis of an ordinary machine, there is a proportional relation between the rotation angle of the motor and the position of the rotation axis.
In this mechanism, in contrast, the proportion of the amount by which the motor (ball screw) rotates to the position of the rotation axis fluctuates depending on the position of the rotation axis.
A position command is output to operate the servo motor. Also, to detect the rotation angle of the pivot axis (B-axis), the mechanism is in a fully closed configuration in which a detector is mounted to the rotation axis itself.
In this mechanism, the speed of the pivot axis (B-axis) may fluctuate. To decrease fluctuations and keep the actual position gain constant, this function compensates the internal value of the position gain according to the position of the pivot axis (B-axis).

Control method

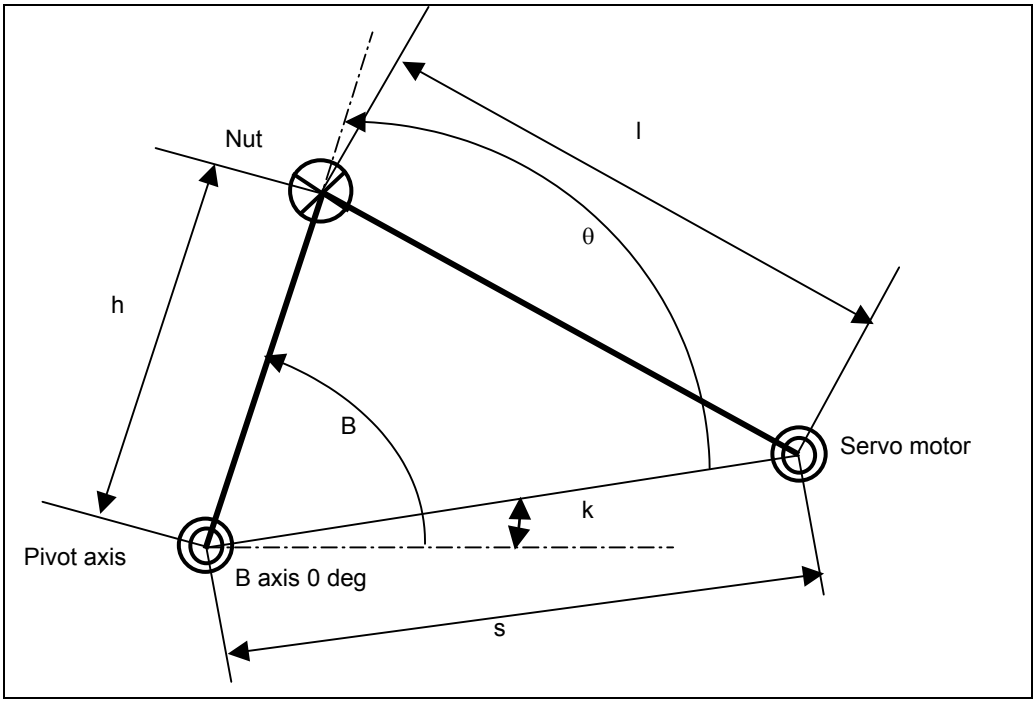


Fig. 21.8 (b)

From Fig. 21.8 (b), the following expression holds:

$$l = \sqrt{h^2 + s^2 - 2sh\cos\theta} \quad \text{.....(1)}$$

The rate of change of l in relation to θ is determined with the expression below.

$$\frac{dl}{d\theta} = \frac{s \cdot h \cdot \sin\theta}{\sqrt{h^2 + s^2 - 2sh \cdot \cos\theta}} \quad \text{.....(2)}$$

The position gain is determined from the relational expression below.

$$G = K\frac{\frac{dl}{dt}}{\frac{d\alpha}{dt}} = K\frac{dl}{d\theta} * \frac{d\theta}{d\alpha} = K\underbrace{\frac{dl}{d\theta} * \frac{\pi}{180}} \quad \text{.....(3)}$$

Gain multiplier

G: Position gain

where α (degree) is the angle of the rotation axis, expressed in radians (θ).

The actual position gain G of the servo motor is defined by the angle represented in the θ-G diagram and the gain multiplier parameter.

NOTE
Before reference position return, the gain multiplier is fixed at 1.

θ-G diagram (relationship between the angle of the rotation axis (θ) and the gain multiplier (G))

Set the angle of the rotation axis in parameters Nos. 14270 to 14279.
Set the gain multiplier in parameters Nos. 14280 to 14289.

Example

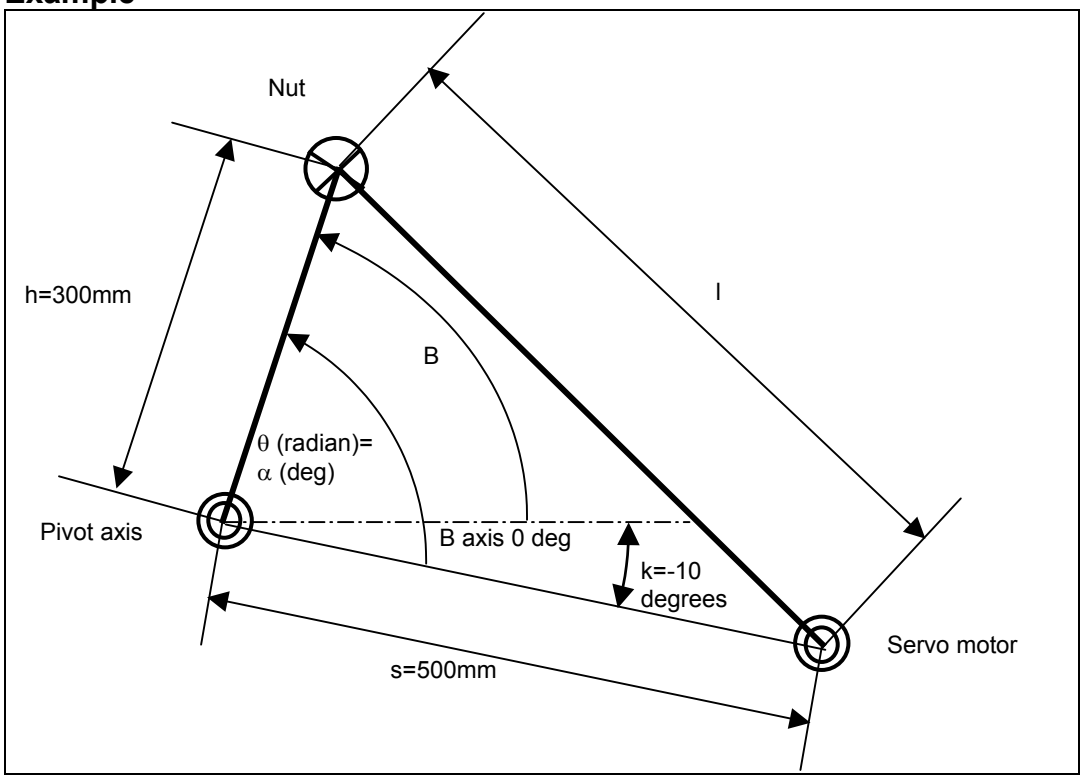


Fig. 21.8 (c)

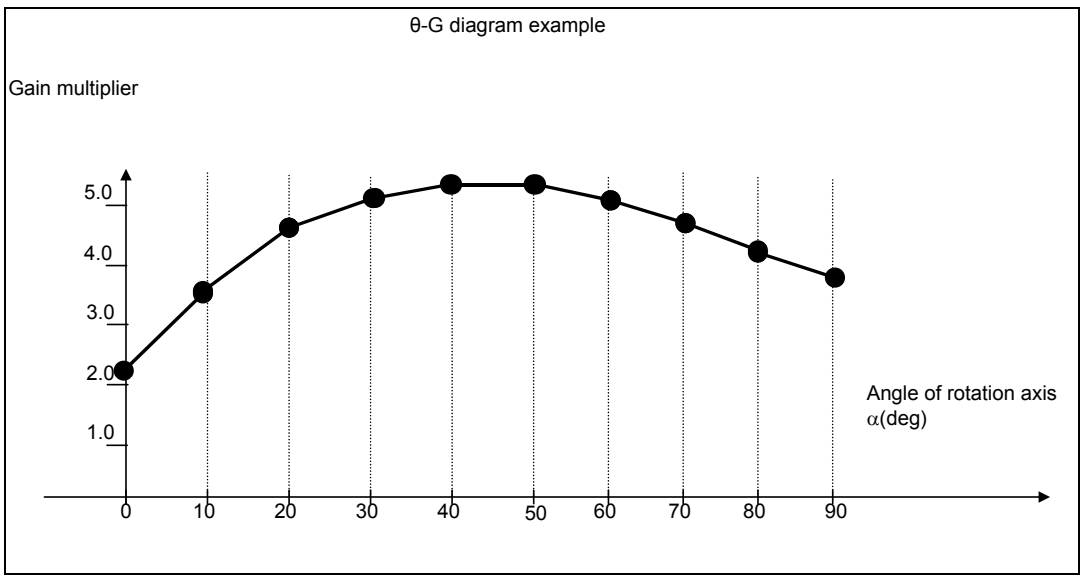


Fig. 21.8 (d)

Angle of the rotation axis
Parameter No. 14270 = 0.0 deg
Parameter No. 14271 = 10.0 deg
Parameter No. 14272 = 20.0 deg
Parameter No. 14273 = 30.0 deg

Parameter No. 14274 = 40.0 deg
Parameter No. 14275 = 50.0 deg
Parameter No. 14276 = 60.0 deg
Parameter No. 14277 = 70.0 deg
Parameter No. 14278 = 80.0 deg
Parameter No. 14279 = 90.0 deg

Gain multiplier

Parameter No. 14280 = 614 (2.2)
Parameter No. 14281 = 1382 (3.7)
Parameter No. 14282 = 1843 (4.6)
Parameter No. 14283 = 2099 (5.1)
Parameter No. 14284 = 2150 (5.2)
Parameter No. 14285 = 2150 (5.2)
Parameter No. 14286 = 2048 (5.0)
Parameter No. 14287 = 1945 (4.8)
Parameter No. 14288 = 1782 (4.5)
Parameter No. 14289 = 1587 (4.1)

Limitation

(1) The servo functions below cannot be used together with this function.
- Dual position feedback function
- Semi-Full error monitor
- Machine speed feedback function
- Vibration damping control function

(2) The functions below cannot be used together with the pivot axis.
- Cs Contour Control
- Threading/Feed per revolution by servo motor
- Temporary absolute coordinate setting

(3) The decision on the pivot axis must be set in bit 0 (PVT) of parameter No. 3456.
(4) Composite control of synchronous/composite control cannot be performed.

NOTE
This function is an optional function. The pivot axis control option is required.

21.9 CHOPPING FUNCTION

Overview

When contour grinding is performed, the chopping function can be used to grind the side face of a workpiece. By means of this function, while the grinding axis (the axis with the grinding wheel) is being moved vertically, a contour program can be executed to initiate movement along other axes.
In addition, a servo delay compensation function is supported for chopping operations. When the grinding axis is moved vertically at high speed, a servo delay and acceleration/deceleration delay occur. These delays prevent the tool from actually reaching the specified position. The servo delay compensation function compensates for any displacement by increasing the feedrate. Thus, grinding can be performed almost up to the specified position.
There are two types of chopping functions: that specified by programming, and that activated by signal input.

Format

G81.1 Z_ Q_ R_ F_ ;

Z :Upper dead point
(For an axis other than the Z-axis, specify the axis address.)
Q :Distance between the upper dead point and lower dead point
(Specify the distance as an incremental value, relative to the upper dead point.)
R :Distance from the upper dead point to point R
(Specify the distance as an incremental value, relative to the upper dead point.)
F :Feedrate during chopping

G80 ; Cancels chopping

NOTE
For the chopping axis, specify a linear axis. It is not possible to perform a chopping operation on a rotation axis.

Explanation

- Chopping activated by signal input

Before chopping can be started, the chopping axis, reference position, upper dead point, lower dead point, and chopping feedrate must be set using the parameter screen (or the chopping screen).
Chopping is started once chopping start signal CHPST has been set to 1. This signal is ignored, however, during chopping axis movement.
When chopping hold signal *CHLD is set to 0 during chopping, the tool immediately moves to point R. Again setting the chopping hold signal to 1 restarts chopping.
Chopping can also be stopped by setting chopping start signal CHPST to 0, but only when chopping was started by using that signal.

Method of starting chopping	Method of stopping chopping	State
Signal CHPST = 1	Signal CHPST = 0	Stopped
	G80	Stopped
G81.1	Signal CHPST = 0	Not stopped
	G80	Stopped

NOTE
1 Switching to manual mode or suspending automatic operation, by means of feed hold, does not stop chopping.
2 In chopping mode, a chopping axis move command or canned cycle command cannot be specified.
3 If a G81.1 command is specified during chopping started by the signal, chopping is not stopped. If point R, the upper dead point, lower dead point, or chopping feedrate has been modified by using the G81.1 command, chopping is continued, using the modified data.
4 The use of chopping start signal CHPST to start chopping is not enabled immediately after power-on; it is not enabled until the completion of manual reference position return.
5 For the chopping axis, specify a linear axis. It is not possible to perform a chopping operation on a rotation axis.

- Chopping feedrate (feedrate of movement to point R)

From the start of chopping to point R, the tool moves at the rapid traverse rate (specified by parameter No. 1420).

The override function can use either the normal rapid traverse override or chopping feedrate override, one of which can be selected by setting ROV (bit 0 of parameter No. 8360).
When the chopping feedrate override is selected, settings between 110% and 150% are clamped to 100%.

- Chopping feedrate (feedrate of movement from point R)

Between point R, reached after the start of chopping, and the point where the chopping is canceled, the tool moves at the chopping feedrate (specified by parameter No. 8374).
The chopping feedrate is clamped to the maximum chopping feedrate (set with parameter No. 8375) if the specified feedrate is greater than the maximum chopping feedrate.
The feedrate can be overridden by 0% to 150% by applying the chopping feedrate override signal.

- Setting chopping data

Set the following chopping data:

- Chopping axis: Parameter No. 8370
- Reference point (point R): Parameter No. 8371
- Upper dead point: Parameter No. 8372
- Lower dead point: Parameter No. 8373
- Chopping feedrate: Parameter No. 8374
- Maximum chopping feedrate: Parameter No. 8375

All data items other than the chopping axis and maximum chopping feedrate can be set on the chopping screen.

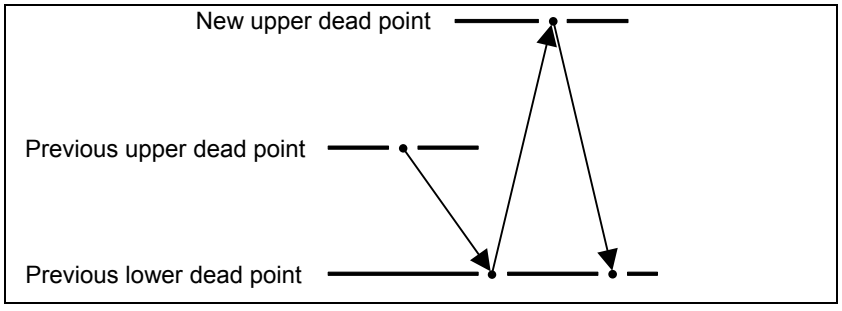
NOTE
Chopping data can be changed using the program command, the parameter screen, and the chopping screen. If, during a chopping operation, the upper dead point, lower dead point, or feedrate for chopping is specified or set, servo delay compensation temporarily stops even if the same value is used. Also, the upper dead point or lower dead point may be exceeded.

- Chopping after the upper dead point or lower dead point has been changed

When the upper dead point or lower dead point is changed while chopping is being performed, the tool moves to the position specified by the old data. Then, chopping is continued using the new data.

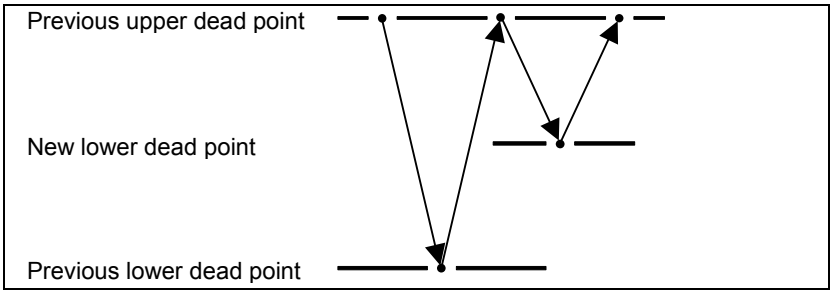
When movement according to the new data starts, the servo delay compensation function stops the servo delay compensation for the old data, and starts the servo delay compensation for the new data.
The following describes the operations performed after the data has been changed.

(1) When the upper dead point is changed during movement from the upper dead point to the lower dead point



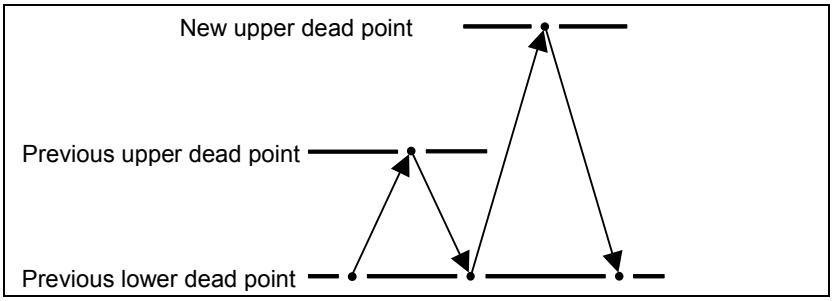
The tool first moves to the lower dead point, then to the new upper dead point.
Once movement to the lower dead point has been completed, the previous servo delay compensation is set to 0, and servo delay compensation is performed based on the new data.

(2) When the lower dead point is changed during movement from the upper dead point to the lower dead point



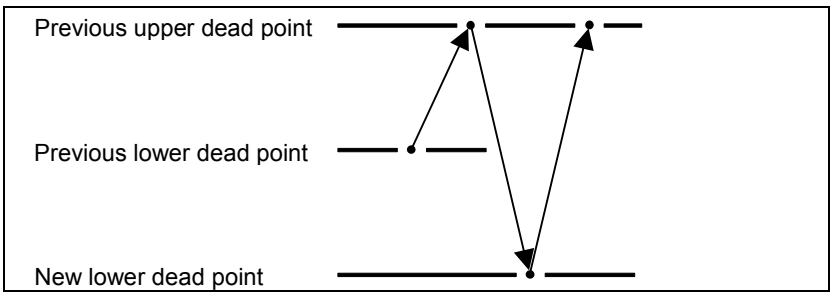
The tool first moves to the previous lower dead point, then to the upper dead point, and finally to the new lower dead point.
Once movement to the upper dead point has been completed, the previous servo delay compensation is set to 0, and servo delay compensation is performed based on the new data.

(3) When the upper dead point is changed during movement from the lower dead point to the upper dead point



The tool first moves to the previous upper dead point, then to the lower dead point, and finally to the new upper dead point.
Once movement to the lower dead point has been completed, the previous servo delay compensation is set to 0, and servo delay compensation is performed based on the new data.

(4) When the lower dead point is changed during movement from the lower dead point to the upper dead point



The tool first moves to the upper dead point, then to the new lower dead point.
Once movement to the upper dead point has been completed, the previous servo delay compensation is set to 0, and servo delay compensation is performed based on the new data.

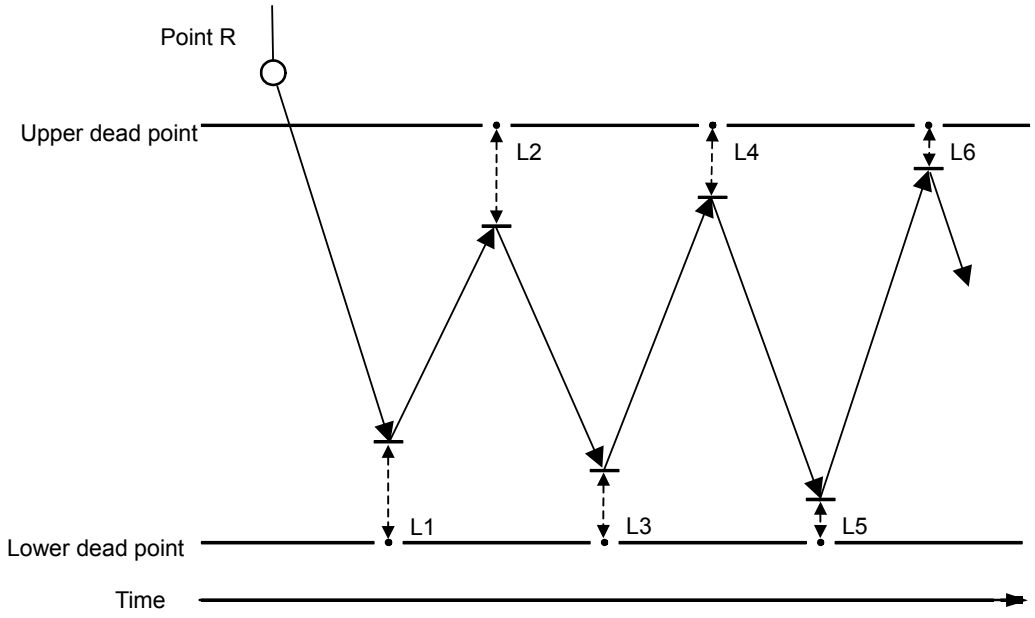
- Servo delay compensation function

When high-speed chopping is performed with the grinding axis, a servo delay and acceleration/deceleration delay occur. These delays prevent the tool from actually reaching the specified position. The control unit measures the difference between the specified position and the actual tool position, and automatically compensates for the displacement of the tool.

To compensate for this displacement, an amount of travel equal to the distance between the upper and lower dead points, plus an appropriate compensation amount, is specified. When a chopping command is specified, the feedrate is determined so that the chopping count per unit time equals the specified count. When the difference between the displacement of the tool from the upper dead point and the displacement of the tool from the lower dead point becomes smaller than the setting of parameter No. 8377, after the start of chopping, the control unit performs compensation.
When compensation is applied, the chopping axis moves beyond the specified upper dead point and lower dead point, and the chopping feedrate increases gradually.
When the difference between the actual machine position and the specified position becomes smaller than in-position width (parameter No. 1826), the control unit stops further compensation, and the tool continue moving at its current feedrate.
A coefficient for the compensation amount for the displacement which has generated by the servo delay and the acceleration /deceleration delay during chopping can be specified in parameter No. 8376.

- If servo delay compensation can cause the chopping speed to exceed the maximum allowable chopping feedrate:

Servo delay compensation during a chopping operation can gradually increase the chopping speed. If the chopping speed is about to exceed the maximum allowable chopping feedrate, it is clamped to the maximum allowable chopping feedrate. In servo delay compensation, the distance specified in a movement command is increased by a compensation amount that matches the distance yet to go before the top and bottom dead points are reached, and the chopping speed is also increased, so that the distance yet to go can be compensated for.
If the chopping speed is clamped to the maximum allowable chopping feedrate, the increase in the distance specified in the movement command stops as soon as the speed is clamped.



Displacement between the tool and the upper dead point: L2, L4, L6
Displacement between the tool and the lower dead point: L1, L3, L5
Compensation starts when:
| L3 - L2 | < (parameter No. 8377)
When the following condition is satisfied, compensation is no longer applied, and the tool continues to move at its current feedrate:
| L6 | < in-position width (parameter No. 1826)

- Acceleration

For the acceleration/declaration along the chopping axis, acceleration/deceleration after cutting feed interpolation is effective.

- Mode switching during chopping

If the mode is changed during chopping, chopping does not stop. In manual mode, the chopping axis cannot be moved manually. It can, however, be moved manually by means of the handle interrupt.

- Reset during chopping

When a reset is performed during chopping, the tool immediately moves to point R, after which chopping mode is canceled.
If an emergency stop or servo alarm occurs during chopping, chopping is canceled, and the tool stops immediately.

- Stopping chopping

The Table 21.9 (a) lists the operations and commands that can be used to stop chopping, the positions at which chopping stops, and the operation performed after chopping stops:

Table 21.9 (a)

Operation/ command	Stop position	Operation after chopping stops
G80	Point R	Canceled
CHPST : “0”	The tool moves to the lower dead point, then to point R	Canceled
*CHLD : “0”	Point R	Restart after *CHLD goes "1"
Reset	Point R	Canceled
Emergency stop	The tool stops immediately	Canceled
Servo alarm	The tool stops immediately	Canceled
PS alarm	The tool moves to the lower dead point, then to point R	Canceled
OT alarm	The tool moves from the upper or lower point to point R	Canceled

- Background editing

When an alarm of background editing or battery alarm is issued, the tool does not stop at point R.

- Single block signal

Even when single block signal SBK is input during chopping, chopping continues.

Limitation

- Workpiece coordinate system

While chopping is being performed, do not change the workpiece coordinate system for the chopping axis.

- PMC axis

When the chopping axis is selected as the PMC axis, chopping is not started.

- Mirror image

While chopping is being performed, never attempt to apply the mirror image function about the chopping axis.

- Move command during chopping

If a move command is specified for the chopping axis while chopping is being performed, an alarm PS5050, “ILL-COMMAND IN G81.1 MODE” is issued.

- Program restart
When a program contains G codes for starting chopping (G81.1) and stopping chopping (G80), an attempt to restart that program results in an alarm PS5050, "ILL-COMMAND IN G81.1 MODE" being output.
When a program that does not include the chopping axis is restarted during chopping, the coordinates and amount of travel set for the chopping axis are not affected after the restart of the program.

- Canned Cycle
While chopping is being performed, do not be specified canned cycle.

- Inch/Metric conversion commands
While chopping is being performed, do not be specified inch/metric conversion commands.

- General purpose retract
While chopping is being performed, do not be performed retraction. The chopping motion does not stop by retraction.

- Stored stroke check
Stored stroke check 1-I (parameter Nos. 1320 and 1321) is only effective during chopping motion.

- Arbitrary angular axis control
Do not set angular axis and perpendicular axis of arbitrary angular axis control to chopping axis.

- Three-dimensional coordinate conversion / Tilted working plane command
While chopping is being performed, do not be specified three-dimensional coordinate conversion / tilted working plane command.

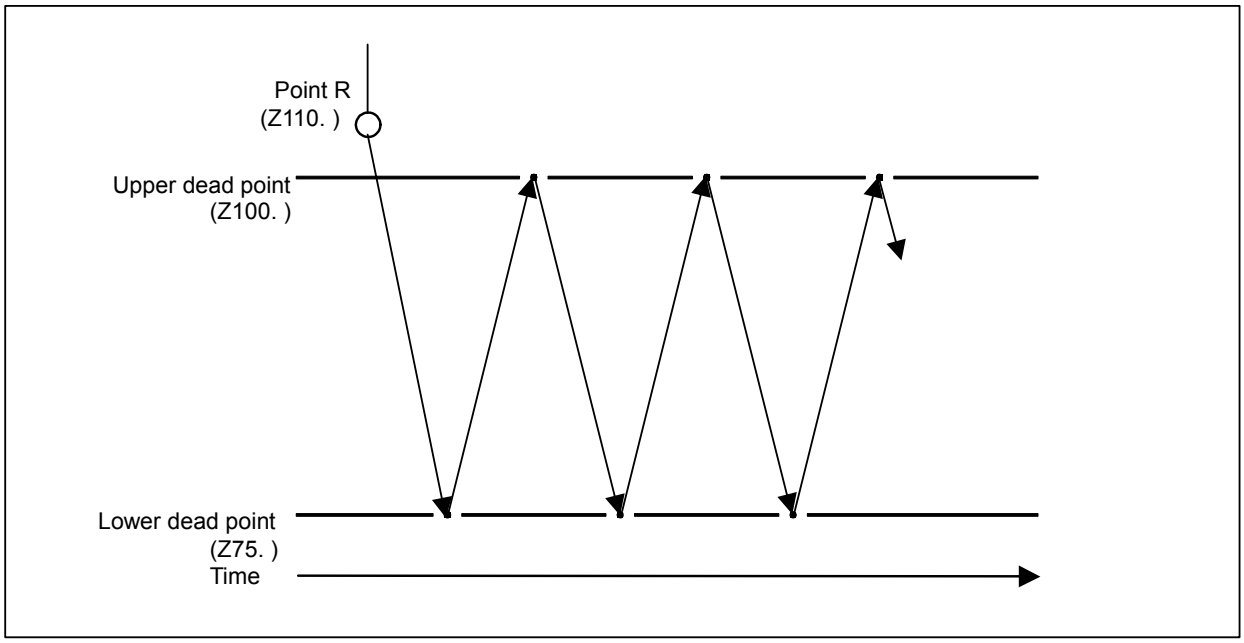
- Cs contour control axis
Do not set Cs contour control axis to chopping axis.

- Composite control
Do not set composite control axis to chopping axis.

- Polygon turning
Do not set control axis for polygon turning to chopping axis.

Example
Example)
G90 G81.1 Z100. Q-25. R10. F3000 ;

- Perform rapid traverse to position the tool to Z110. (point R).
- Then, perform reciprocating movement along the Z-axis between Z100. (upper dead point) and Z75. (lower dead point) at 3000 mm/min. Chopping override is enabled.



To cancel chopping, specify the following command:

G80 ;

- The tool stops at point R.

21.10 SKIP FUNCTION FOR FLEXIBLE SYNCHRONOUS CONTROL

Outline

This function enables the skip or high-speed skip signal (in the following explanation, these signals are collectively called skip signal) for the slave axis that is moved by command of the master axis in the flexible synchronous control mode.

This function has features such as the following:

- If a skip signal is input while a skip command for flexible synchronous control block is being executed, this block does not terminate until the specified number of skip signals have been input.
- The machine coordinates assumed when skip signals are input and the number of input skip signals are stored in specified custom macro variables.
- The total number of the skip signal inputs is stored in another specified custom macro variable.

This function is an optional function.

Format

Mxx ; Flexible synchronous control mode on

G31.8 G91 α 0 P_ Q_ R_ ; Skip command for flexible synchronous control

α : Specify the slave axis. The instruction value must be 0.
P_: The top number of the consecutive custom macro variables in which the machine coordinate positions of the slave axis at the skip signal inputs are stored.
Q_: The maximum allowable number of the skip signal inputs. (Range of command value: 1 to 512)
R_: The number of the custom macro variables in which the total number of the inputs is stored.
This data is usually the same as the value specified by Q.
Therefore this is not necessarily specified. Specify it to check the number of skip signal inputs.

G31.8 is a one-shot G code.
During the execution of the G31.8 block, the machine coordinate positions of the slave axis at the skip signal inputs are stored in the consecutive custom macro variables where the top number of the variables is specified by P and the maximum allowable number of the skip signal inputs is specified by Q.
Also, this total number of the skip signal inputs is stored in the variable specified by R.

Example)

Mxx	Flexible synchronous control mode on
X--	
Y--	
G31.8 G91 A0 P100 Q30 R1	Skip command for flexible synchronous control

After 30 times of skip signal inputs, 30 machine coordinate positions of the A axis are stored respectively in the consecutive custom macro variables #100 to #129.
The total number of skip signal inputs is stored in the custom macro variable #1.

NOTE
1 In the G31.8 block, only one slave axis should be commanded. When more than two slave axes are specified, the alarm (PS1152) occurs.
2 If G31.8 is commanded out of the flexible synchronous control mode (Flexible synchronous control mode select signal switching accepted signal (MFSYNA, MFSYNB, MFSYNC or MFSYND) is “0”), the alarm (PS1152) occurs.
3 If P is not specified, the alarm (PS1152) occurs.
4 If R is not specified, the number of skip signal inputs is not stored in the custom macro variables
5 The number of custom macro variables specified in P and R must be the existing ones. If any nonexistent variable is specified, the alarm (PS0115) occurs. If a variable shortage occurs, the alarm (PS0115) occurs, too.
6 Whether to use conventional skip signals or high-speed skip signals with this function can be specified with the parameter HSS (No.6200#4). When high speed skip is selected, specify which high-speed signals to enable with setting parameter 9S1 to 9S8 (No.6208#0 to #7).
7 The accumulated pulses and positional deviation due to acceleration/deceleration is considered and compensated when storing the machine coordinate positions to the custom macro variables.
8 The option of custom macro is necessary to use this function.

21.11 CHOPPING FUNCTION BY FLEXIBLE SYNCHRONOUS CONTROL

Overview

This function enables the chopping of simultaneous 2-axis control by using a flexible synchronous control with the chopping.
It is possible to synchronize an axis with a chopping axis. This function is suitable for the taper hole processing by a grinder etc.

Explanation

Override of gear ratio for flexible synchronization

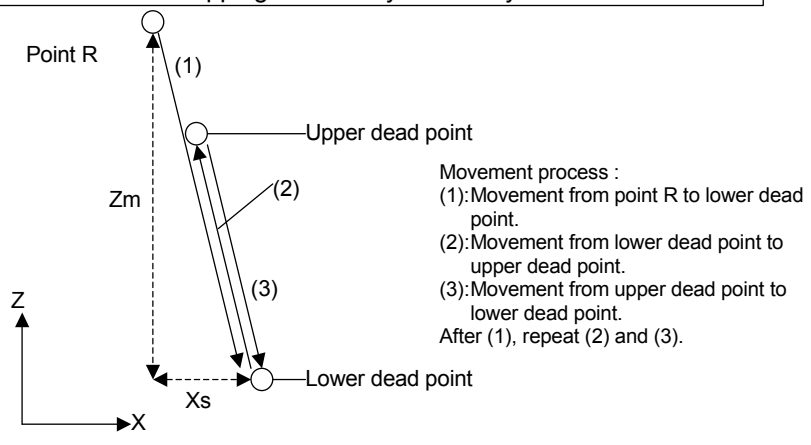
This function enables to override the commanded gear ratio of flexible synchronous control.

Program example

Axis configuration: X,Y,Z,C, and U
Setting of flexible synchronous control :
Master axis : Z, slave axis : U,
M code for turning on the flexible synchronization control mode : M50
M code for turning off the flexible synchronization control mode : M51

```
G90 G00 X100.0 Y75.0 Z120.0 C0 U0 ;  ..... Move to the start point.
M50 ; ..... Start Z-U synchronization.
G81.1 Z100.0 Q-25.0 R10.0 F3000. ; ..... Start chopping.
……
……
G80 ; ..... Cancel chopping.
M51 ; ..... Cancel Z-U synchronization.
```

Movement of chopping function by flexible synchronous control



When flexible synchronization is applied to Z axis (master) and X axis (slave), Amount of travel Xs of X axis is as follows:

Xs = Zm × Gear ratio for flexible synchronization (parameter No.5680 to No.5693)
× override signal of gear ratio for flexible synchronization

Notes

1 The parameter setting and notes for flexible synchronous control and chopping are applied to this function.

2 Flexible synchronous control is started and canceled by the flexible synchronous control mode select signal. To start or cancel flexible synchronization during automatic operation, control the select signal using the M code set in the relevant parameter.
3 Chopping can be specified using the following two ways: Specifying chopping in the program and using the chopping start signal. Chopping started by specifying it in the program cannot be canceled by inputting the signal.
4 Be sure to start or cancel chopping in flexible synchronous control for this function. (See "Program example".)
5 Before synchronization starts, the tool must be positioned along the slave axis.
6 When the servo delay compensation function during chopping operation is used with this function, the compensation amount for the master axis is multiplied by the gear ratio of flexible synchronization, and the result is output to the slave axis. If you want to adjust the compensation amount for the slave axis, adjust the gear ratio using override signals of gear ratio for flexible synchronization.
7 When this function is used for a rotation axis, the servo delay compensation function during chopping operation cannot be used.

21.12 HIGH PRECISION OSCILLATION FUNCTION

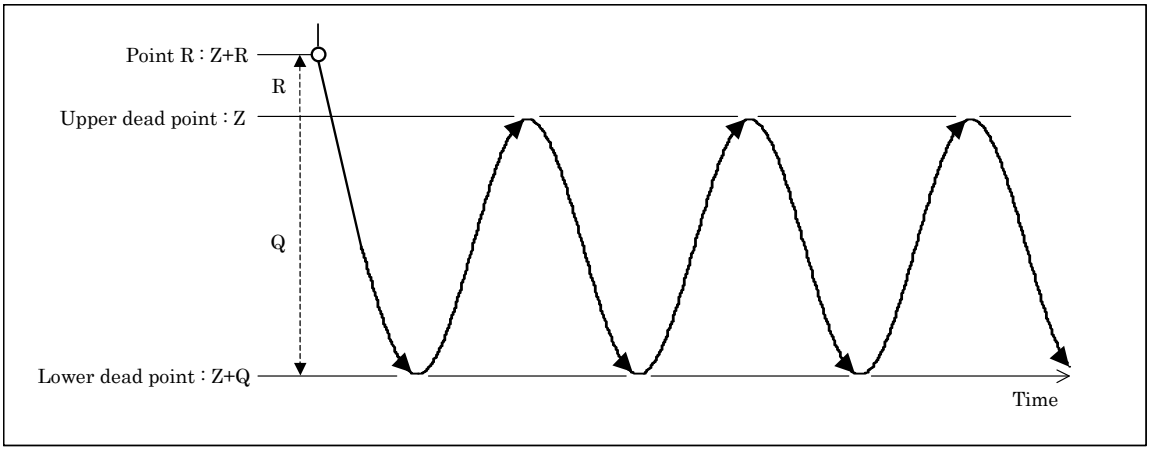
Overview

In this function, the feedrate of oscillation axis (equivalent to a chopping axis that is moved vertically and repeatedly for grinding) changes along sine curve. This function is effective to improve, the accuracy of movement between upper dead point and lower dead point.
Moreover, look-ahead feed forward function can be applied to oscillation motion, then higher accuracy can be achieved even if oscillation feedrate or upper or lower dead point is changed.

This function is optional function.

Format

G81.1 Z_ Q_ R_ F_ ;
Z : Upper dead point (In case that an axis is other than the Z-axis, specify the axis address.)
Q : Distance between the upper dead point and lower dead point
(Specify the distance as an incremental value from the upper dead point.)
R : Distance from the upper dead point to point R
(Specify the distance as an incremental value from the upper dead point.)
F : Oscillation base feedrate
G80 ; Cancels oscillation



If the addresses of Z, Q, R, or F are omitted, the oscillation motion is performed by the value of parameters. On the other hand, the value of parameters is replaced by the value commanded for each address.

Z (Axis address)	Parameter (No.8370)	: Oscillation axis number = Axis number.
	Parameter (No.8372)	: Upper dead point = Value of address Z.
Q	Parameter (No.8373)	: Lower dead point = Value of address Q + value of address Z (If address Z is omitted, the value is parameter (No.8372)).
R	Parameter (No.8371)	: Point R = Value of address R + value of address Z (If address Z is omitted, the value is parameter (No.8372)).
F	Parameter (No.8374)	: Oscillation base feedrate = Value of address F.

Explanation

- Oscillation activated by signal input

Before oscillation motion is started, oscillation axis, reference position, upper dead point, lower dead point, and oscillation base feedrate must be set on the parameter screen or the oscillation screen. Oscillation motion is started once oscillation start signal CHPST has been set to "1". This signal is ignored, however, while oscillation axis is moving.

When oscillation hold signal *CHLD is set to "0" during oscillation motion, the tool immediately moves to point R and stops. Setting the oscillation hold signal to "1" restarts oscillation motion. Oscillation motion can also be stopped by setting oscillation start signal CHPST to "0", but only when oscillation motion was started by using that signal.

Method of starting oscillation	Method of cancelling oscillation	State
Signal CHPST ="1"	Signal *CHLD = "0"	It stops. (Suspending state)
	Signal CHPST = "0"	It stops. (Cancel state)
	G80 command	It stops. (Cancel state)
G81.1 command	Signal *CHLD = "0"	It stops. (Suspending state)
	Signal CHPST = "0"	It doesn't stop.
	G80 command	It stops. (Cancel state)

NOTE

1 Switching to manual mode or suspending automatic operation, by means of feed hold, does not stop oscillation motion.
2 If a move command is specified for the oscillation axis while oscillation is being performed, an alarm (PS5050) is issued.
3 If a G81.1 command is specified during oscillation started by the signal, oscillation motion is not stopped. If point R, upper dead point, lower dead point, or oscillation feedrate has been modified by using the G81.1 command, oscillation is continued but using the modified data.
4 The use of oscillation start signal CHPST <Gn051.6> to start oscillation motion is not enabled immediately after power-on; it is not enabled until the completion of manual reference position return.
5 The M codes which are used to set the oscillation start signal CHPST <Gn051.6> to "1" or "0" should be specified as buffering mask type M code (parameter Nos. 3411 to 3432).

- Oscillation feedrate (feedrate of movement to point R)

From the start of oscillation motion to reaching point R, the tool moves at the rapid traverse rate (specified by parameter No.1420).

The override function can be used for either the normal rapid traverse rate or oscillation feedrate, one of which can be selected by setting ROV (bit 0 of parameter No.8360). When the oscillation feedrate is overridden, settings between 110% and 150% are clamped to 100%.

- Oscillation feedrate (feedrate of movement from point R)

From point R, reached after the start of oscillation motion, to the center point between upper and lower dead points, the tool moves at the oscillation base feedrate (F). Afterwards, between the center point and the point where the oscillation motion is canceled, the tool moves with sine curve feedrate by Exp. 1.

$$f(t)_{[mm/\min]} = k \times \mathrm{F}_{[mm/\min]} \times \sin\left(\frac{2}{|\mathrm{Q}|_{[1/mm]}} \times \frac{k \times \mathrm{F}}{60}_{[mm/\sec]} \times \frac{180}{\pi}_{[\deg/rad]} \times t_{[\sec]} \right) \quad \ldots \text{Exp. 1}$$

F(t) : Sine curve feedrate [mm/min]
F : Oscillation base feedrate [mm/min]
Q : Distance between the upper dead point and lower dead point [mm]
k : Oscillation override (0.0 (0%) to 1.5 (150%))
t : Time [sec]

The value (*k*F) of the product of oscillation base feedrate and oscillation override is clamped to the maximum oscillation feedrate (parameter No.8375) if the value (*k*F) is greater than the maximum oscillation feedrate. However, even if the value (*k*F) is not greater than maximum oscillation feedrate, it is clamped by the feedrate never exceeds the maximum allowable acceleration rate for acceleration/deceleration (parameter No.25652) during oscillation motion.

- Setting oscillation data

Set the following oscillation data:

Oscillation axis……Parameter (No.8370)
Reference point (point R)……Parameter (No.8371)
Upper dead point……..Parameter (No.8372)
Lower dead point……Parameter (No.8373)
Oscillation base feedrate……..Parameter (No.8374)
Maximum oscillation feedrate… Parameter (No.8375)

All data items other than the oscillation axis and maximum oscillation feedrate can be set on the oscillation screen. Moreover, the value of parameters is replaced by the value of G81.1 command.

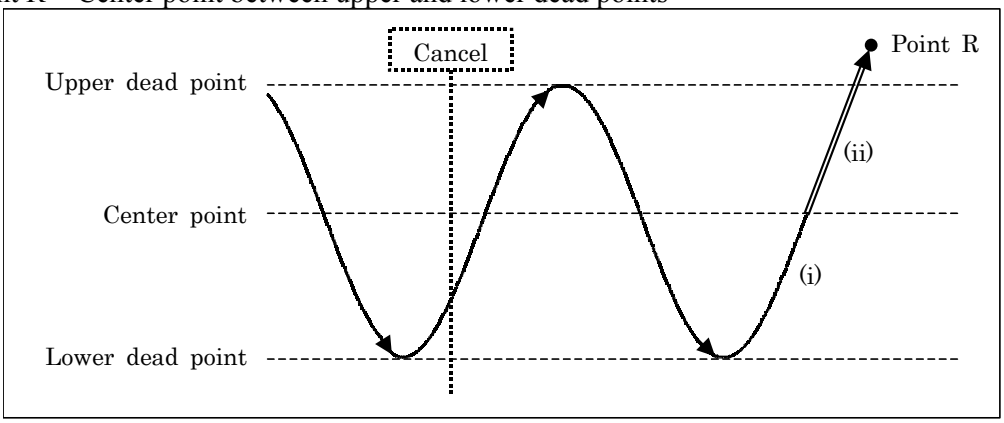
- Stopping oscillation

The following table lists the operations and commands that can be used to stop oscillation motion, the positions at which oscillation stops, and the operation performed after oscillation stops:

Operation/ command	Stop position	Operation after stops
G80 command	It follows parameter OST (No.25651#0). OST=0 : Point R OST=1 : Decelerated and stopped	Canceled
Reset		
PS, OT alarm		
CHPST: "0"	It follows parameters OST, SGS (No.25651#0,#2). OST=0 or SGS=0 : Point R OST=1 and SGS=1 : Decelerated and stopped	Canceled
*CHLD: "0"	It follows parameters OST, SGS, HST (No.25651#0,#2,#3). OST=0, SGS=0, or HST=0 : Point R OST=1, SGS=1, and HST=1 : Decelerated and stopped	Restart after *CHLD goes "1"
Emergency stop	The tool stops immediately.	Canceled
Servo alarm		

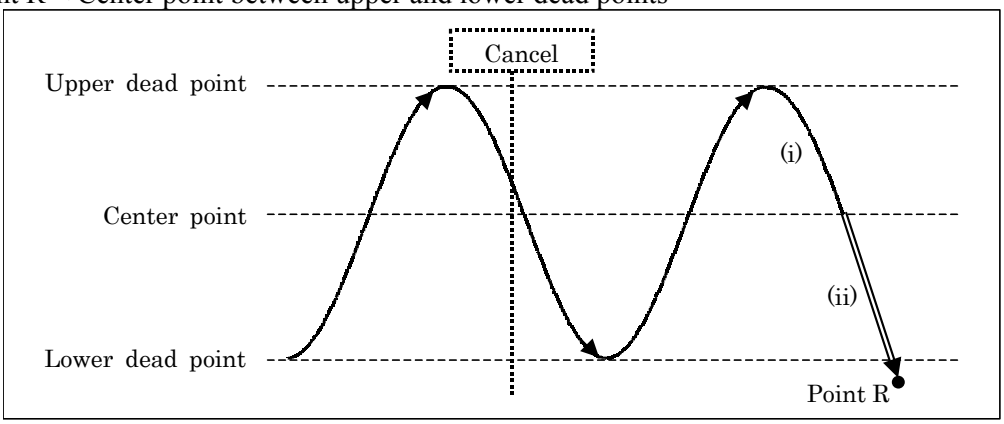
Example in the setting that stop position is point R. (In case of other than suspending by signal *CHLD and OT alarm.)

(1) Point R > Center point between upper and lower dead points



After cancel command, oscillation motion is continued until oscillation axis passes center point between upper and lower dead points via next lower dead point (i).
Afterwards, the axis moves to point R by linear interpolation (ii).

(2) Point R < Center point between upper and lower dead points



After cancel command, oscillation motion is continued until oscillation axis passes center point between upper and lower dead points via next upper dead point (i).
Afterwards, the axis moves to point R by linear interpolation (ii).

In the case of suspending by signal *CHLD or cancel by OT alarm, oscillation axis moves to point R at once.

- Movement from point R to the first dead point

The first dead point is decided by the position of point R.

Point R	The first dead point
Point R is on upper dead point side from center point or is on the center point.	Lower dead point
Point R is on lower dead point side from center point.	Upper dead point

- Omission of movements to point R and to the center point

By setting 1 to parameters OST (No.25651#0) and FFS (No.25651#1), if oscillation motion is started by G81.1 command, movements to point R and to the center point can be omitted, and oscillation motion can be immediately started. In this case, the first dead point is as follows.

State before G81.1 command	The first dead point
In case of the first command of oscillation motion after the power is turned on or in the state of reset.	Lower dead point
In the state that command of cancel was specified on the way to lower dead point at last oscillation motion.	

State before G81.1 command	The first dead point
In the state that command of cancel was specified on the way to upper dead point at last oscillation motion.	Upper dead point

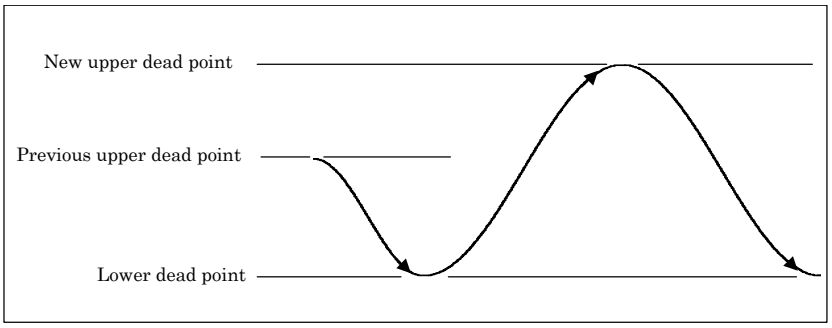
- Acceleration/deceleration

Linear acceleration/deceleration by parameter No.25653 is applied to the movement from point R to center point or the motion till stop.
Moreover, when override is changed during oscillation motion, the acceleration/deceleration set by parameter No.25653 is effective.
Acceleration/deceleration of sine curve feedrate control is enabled on oscillation motion.

- Oscillation after the upper dead point or lower dead point has been changed

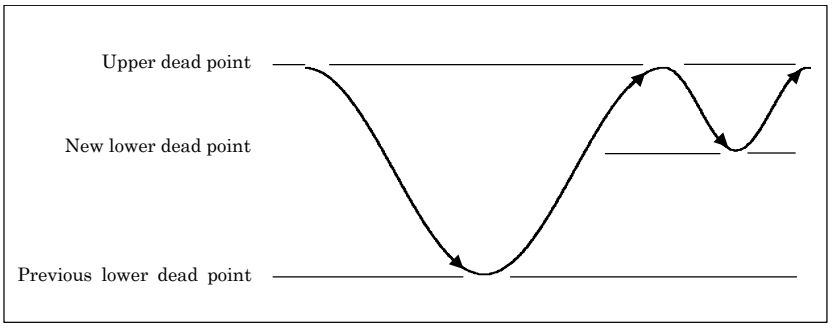
When the upper dead point or lower dead point is changed while oscillation motion is being performed, the tool moves to the dead points specified by the old data. Then, oscillation motion is changed to the new one using the new data.
The following describes the operations performed after the data has been changed.

(1) When the upper dead point is changed during movement from the upper dead point to the lower dead point



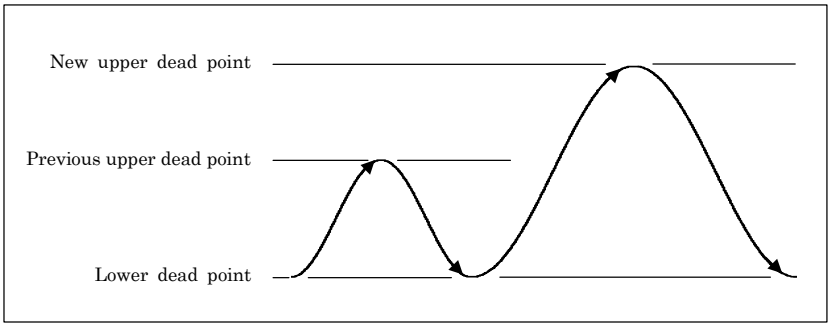
The tool first moves to the lower dead point, then to the new upper dead point.

(2) When the lower dead point is changed during movement from the upper dead point to the lower dead point



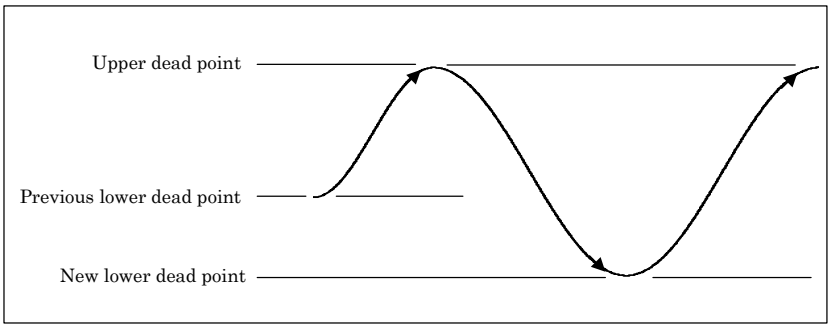
The tool first moves to the previous lower dead point, then to the upper dead point, and finally to the new lower dead point.

(3) When the upper dead point is changed during movement from the lower dead point to the upper dead point



The tool first moves to the previous upper dead point, then to the lower dead point, and finally to the new upper dead point.

(4) When the lower dead point is changed during movement from the lower dead point to the upper dead point



The tool first moves to the upper dead point, then to the new lower dead point.

- Look-ahead feed forward function

Look-ahead feed forward function can be effective for oscillation motion.

- Mode switching during oscillation motion

If the mode is changed during oscillation motion, oscillation motion does not stop. In manual mode, the oscillation axis cannot be moved manually. It cannot also be moved manually by means of the handle interrupt.

- Single block signal

Even when single block signal SBK is input during oscillation motion, oscillation continues.

- Interlock

If interlock is effective to the oscillation axis during oscillation motion, the oscillation axis stops immediately.

- Chopping function

This oscillation function cannot be used together with chopping function. The oscillation function becomes effective by setting the parameter SSO (No.25650#0) to 1.

Limitation

- Workpiece coordinate system

While oscillation is being performed, do not change the workpiece coordinate system for the oscillation axis.

- PMC axis

When the oscillation axis is operating as the PMC axis, oscillation is not started.

- Mirror image

Never attempt to apply the mirror image function about the oscillation axis.

- Move command during oscillation motion

If a move command is specified for the oscillation axis while oscillation motion is being performed, an alarm (PS5050) "During chopping, a move command has been issued for the chopping axis." is issued.

- Canned Cycle

While oscillation is being performed, do not be specified canned cycle.

- Program restart / Quick program restart

When a program contains G codes for starting oscillation (G81.1) and stopping oscillation (G80), an attempt to restart that program results in an alarm (PS5050) being output.
When a program that does not include the oscillation axis is restarted during oscillation motion, the coordinates and amount of travel set for the oscillation axis are not affected after the restart of the program.

- Rotation axis

Oscillation operation cannot be used with rotation axis (type-A).

- Inch/Metric conversion commands

While oscillation is being performed, do not be specified inch/metric conversion commands.

- General purpose retract

While oscillation is being performed, do not be performed retraction. The oscillation motion does not stop by retraction.

- Stored stroke check

Stored stroke check 1-I (parameter Nos. 1320 and 1321) is only effective during oscillation motion.

- Arbitrary angular axis control

Do not set angular axis and perpendicular axis of arbitrary angular axis control to oscillation axis.

- Three-dimensional coordinate conversion / Tilted working plane command

While oscillation is being performed, do not be specified three-dimensional coordinate conversion / tilted working plane command.

- Cs contour control axis

Do not set Cs contour control axis to oscillation axis.

- Composite control

Do not set composite control axis to oscillation axis.

- Polygon turning

Do not set control axis for polygon turning to oscillation axis.

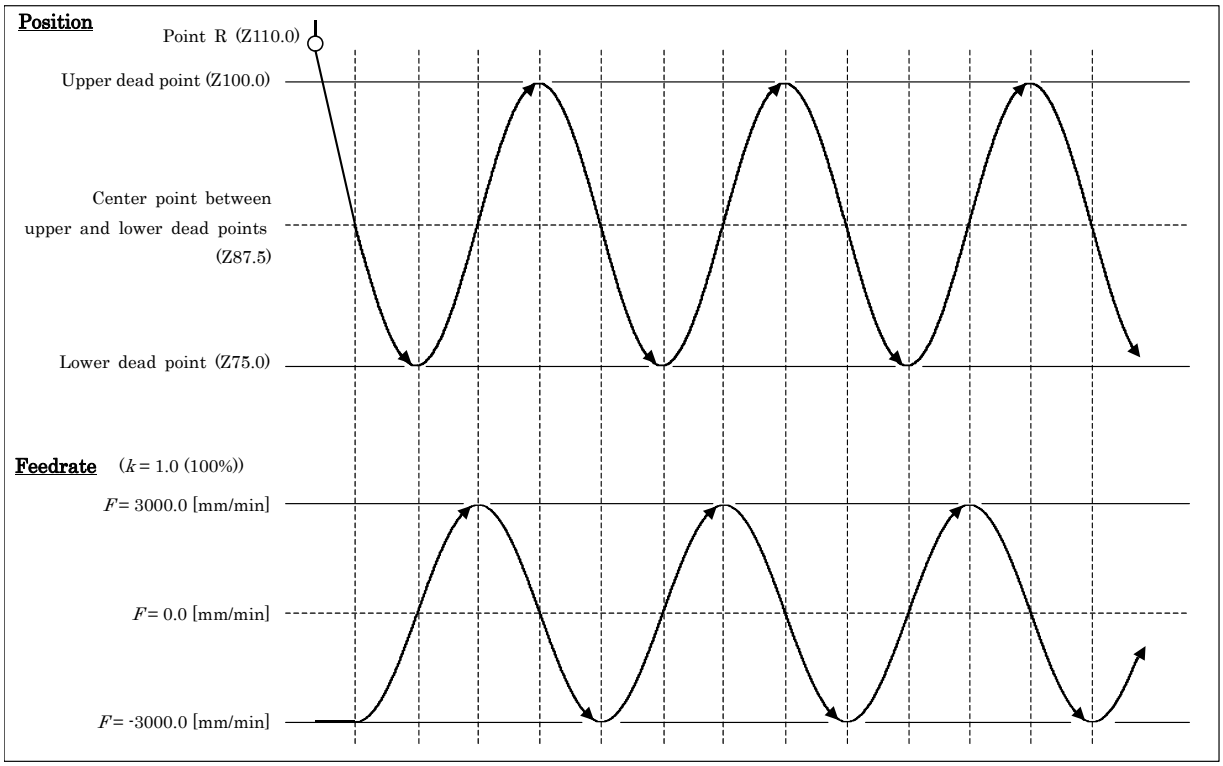
Example

To start oscillation, specify the following command:

```
G90 G81.1 Z100. Q-25. R10. F3000. ;
```

- The tool moves to point R (Z+R = Z110.0) with rapid traverse feedrate.
- Next, move to center point (Z87.5) between upper dead point (Z100.0) and lower dead point (Z+Q = Z75.0) at oscillation base feedrate (F3000.0[mm/min]).

- Then, perform repeated movement along the Z-axis between the upper dead point and the lower dead point with sine curve feedrate F by Exp. 1. Oscillation override k is enabled.



To cancel oscillation motion, specify the following command:

G80 ;

Motion of cancel is as follows.

Parameter OST (No.25651#0) is

0: The tool moves to point R and stops.

1: The tool decelerates and stops.

22 5-AXIS MACHINING FUNCTION

Chapter 22, "5-AXIS MACHINING FUNCTION", consists of the following sections:

22.1 TOOL CENTER POINT CONTROL 682
22.2 HIGH-SPEED SMOOTH TCP 739
22.3 EXPANSION OF AXIS MOVE COMMAND IN TOOL CENTER POINT CONTROL 764
22.4 TOOL POSTURE CONTROL 766
22.5 CUTTING POINT COMMAND 776
22.6 TILTED WORKING PLANE INDEXING 786
22.7 INCLINED ROTARY AXIS CONTROL 847
22.8 3-DIMENSIONAL CUTTER COMPENSATION 850
22.9 EXPANSION OF THE WAY TO SET 5-AXIS MACHINING FUNCTION PARAMETERS 900
22.10 MACHINE CONFIGURATION SELECTING FUNCTION 903

22.1 TOOL CENTER POINT CONTROL

Overview

On a 5-axis machine having two rotary axes that turn a tool or table, this function performs tool length compensation constantly, even in the middle of a block, and exerts control so that the tool center point moves along the specified path. (See Fig. 22.1 (a).)

There are three different types of 5-axis machines - <1> those that rotate the tool only, <2> those that rotate the table only, and <3> those that rotate both the tool and table. (See Fig. 22.1 (d).)

This function is intended to perform machining on such 5-axis machines having rotary axes that turn a tool or table as well as three orthogonal axes (X-, Y-, and Z-axes) by accomplishing tool length compensation while changing the attitude of the tool. It enables the tool center point to move along the specified path even if the tool's direction changes with respect to the workpiece.

A coordinate system used for programming the tool center point control is called the programming coordinate system.

A coordinate system fixed on the table can be used as the programming coordinate system, which makes CAM programming easy. This coordinate system is particularly called the table coordinate system.

A workpiece coordinate system fixed on a machine coordinate system can be employed as the programming coordinate system as well.

In any case, the cutting speed can be controlled easily because the tool center point moves at a specified speed with respect to the table (workpiece).

The commands that can be issued during tool center point control are positioning (G00), linear interpolation (G01), circular interpolation (G02, G03), and helical interpolation (G02, G03).

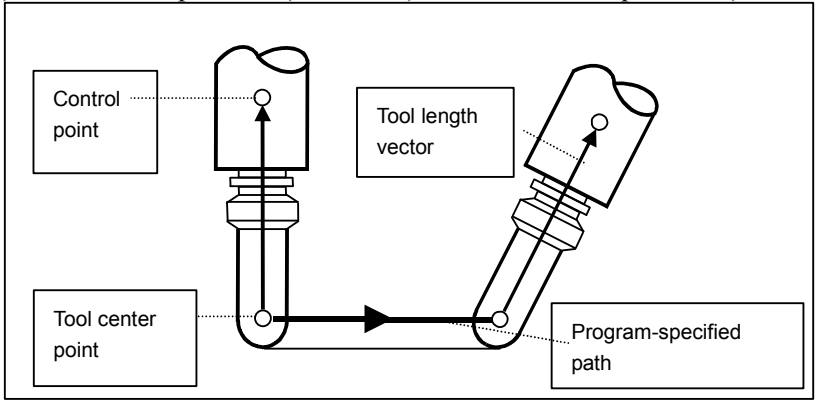


Fig. 22.1 (a) Path of the tool center point

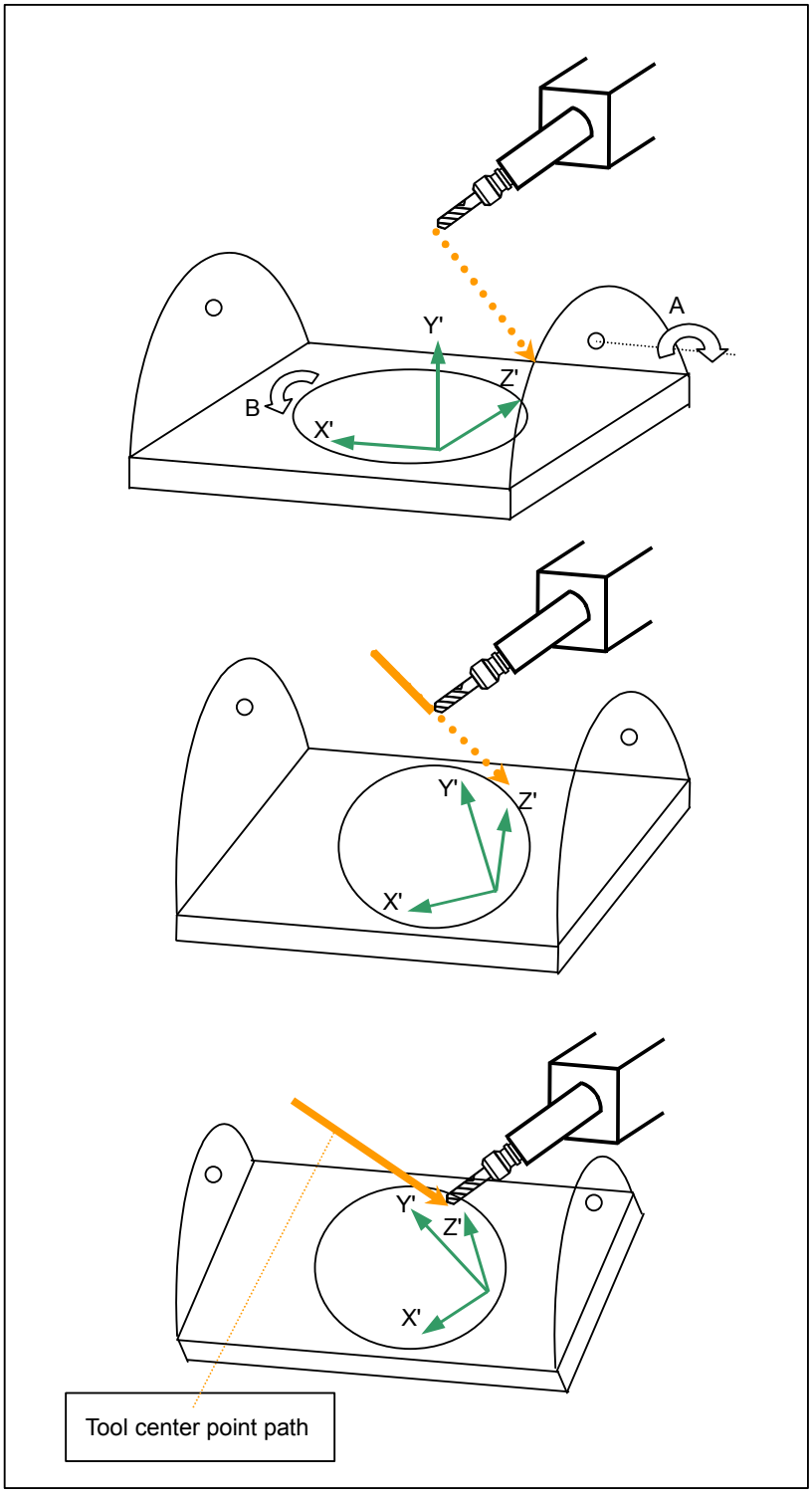


Fig. 22.1 (b) Path of the tool center point

When a coordinate system fixed on the table is used as the programming coordinate system, programming can be performed without worrying about the rotation of the table because the programming coordinate system does not move with respect to the table, although the position and direction of the workpiece fixed on the table change due to its rotation. When a straight line is specified, the tool center point moves along a straight path with respect to the workpiece as instructed. (See Fig. 22.1 (b).)

By setting the relevant parameter, the workpiece coordinate system can also be employed as the programming coordinate system. In this case, as the table turns, the position and direction of the workpiece fixed on the table change with respect to the programming coordinate system. It is therefore necessary to take into account the rotation of the table when specifying the end point. In this case, too, when a straight line is specified, the tool center point moves along a straight path with respect to the workpiece as instructed.
Fig. 22.1 (c) illustrates how linear interpolation is accomplished with a composite type machine, showing the relationship between the case when a table-fixed coordinate system is used as the programming coordinate system and the case when the workpiece coordinate system is used. If linear interpolation is specified in this function mode, speed control is exerted in such a way that the tool center point moves at a specified speed with respect to the workpiece.

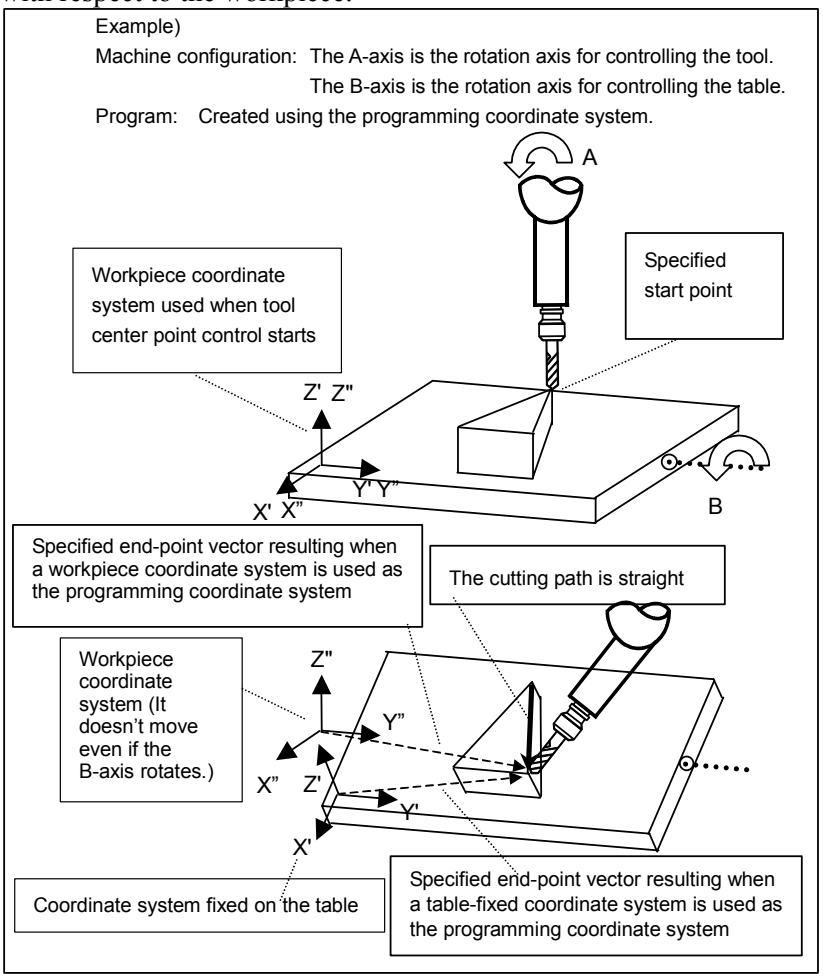


Fig. 22.1 (c) Linear interpolation with a composite type machine

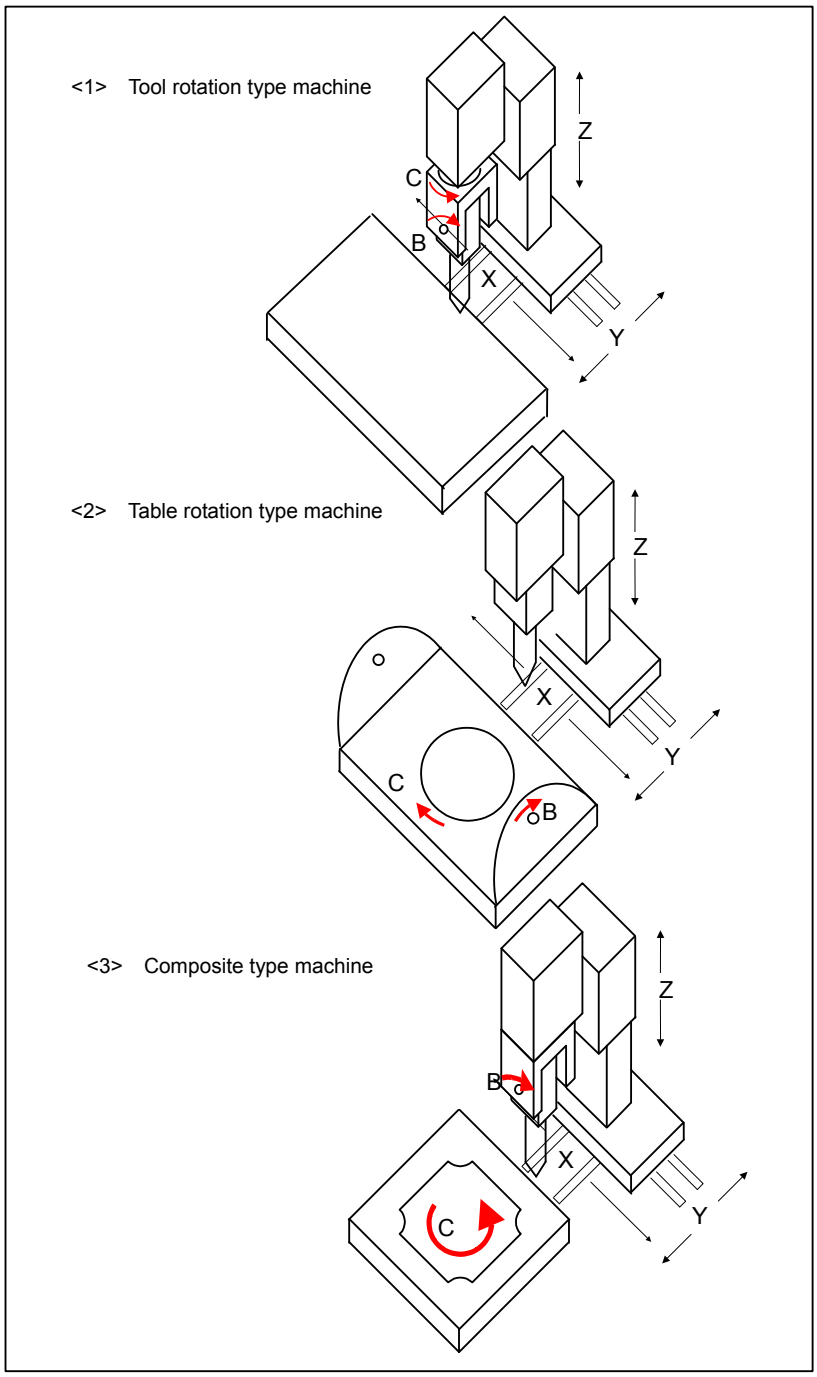


Fig. 22.1 (d) Three types of 5-axis machine

Even if the rotary axis that controls the tool does not intersect the one that controls the table, this function can still be used.

There are two types, as described below, one of which is used depending on how the direction of the tool axis is specified.

(1) Type 1
The block end point of the rotary axes is specified (e.g. A, B, C).
The CNC performs tool length compensation by the specified amount in the tool axis direction that is calculated from the specified position of the rotary axes and exerts control so that the tip of the tool moves along the specified path.

(2) Type 2
The direction of the tool axis (I, J, K) at the block end point, as seen from the coordinate system fixed on the table, is specified, instead of the position of the rotary axes.
The CNC calculates an end point of the rotary axes where the tool will face the specified direction, performs tool length compensation by the specified amount in the tool axis direction that is calculated from the position of the rotary axes, and exerts control so that the tip of the tool moves along the specified path.

Format

- Positioning and linear interpolation for tool center point control (type 1)

G43.4 IP_ α_ β_ H_ ; Starts tool center point control (type 1).

IP_ α_ β_ ;

:

IP : In the case of an absolute programming, the coordinate value of the end point of the tool tip movement
In the case of an incremental programming, the amount of the tool tip movement

α, β : In the case of an absolute programming, the coordinate value of the end point of the rotary axes
In the case of an incremental programming, the amount of the rotary axis movement

H : Tool offset number

Movement to the position specified by the G43.4 block (startup) does not constitute tool center point control. Only tool length compensation is performed.
As for rotary axes, either table rotation axes or tool rotation axes are specified.

While performing interpolation for the rotary axes, the CNC controls the control points so that the tool center point moves along a straight line with respect to the table (workpiece). The end of the tool center point comes to the point specified on the programming coordinate system.

- Positioning and linear interpolation for tool center point control (type 2)

G43.5 IP_ H_ Q_ ; Starts tool center point control (type 2).

IP_ I_ J_ K_ ;

:

IP : In the case of an absolute programming, the coordinate value of the end point of the tool tip movement
In the case of an incremental programming, the amount of the tool tip movement

I, J, K : Tool axis direction at the block end point as seen from the programming coordinate system

H : Tool offset number

Q : Inclination angle of the tool (in degrees)

Movement to the position specified by the G43.5 block does not constitute tool center point control. Only tool length compensation is performed.
No rotary axes are specified. Instead, the direction of the tool end point is specified as I, J, K, as seen from the programming coordinate system (the one fixed on the table when G43.5 is specified).
In the case of a tool rotation type machine, I, J, K can be specified in the G43.5 block. But in the case of a table rotation type or composite type machine, I,J,K cannot be specified in the G43.5 block. Specifying I,J,K in the G43.5 block with a table rotation type or composite type machine causes alarm PS5421.

The significant digit number of decimals on the tool directional vector I, J, and K is up to 11 digits. However, because total digit number of integer and decimal is up to 12 digits, Maximum command digit number of integer depends on the increment system as indicated in following table.

	Increment system				
	IS-A	IS-B	IS-C	IS-D	IS-E
Maximum command digit number of integer	10	9	8	7	6

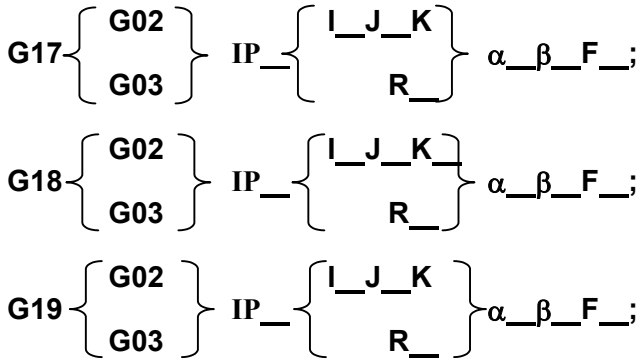
While performing interpolation for the rotary axes, the CNC controls the control points so that the tool center point moves along a straight line with respect to the table (workpiece). The end of the tool center point comes to the point specified on the programming coordinate system.

⚠ CAUTION

1 If one or two of the I, J, and K values are omitted, the omitted value or values are considered to be 0.
2 In a block in which I, J, and K are all omitted, the compensation vector of the preceding block is used.
3 This block can be used only when the programming coordinate system is fixed on the table (when the bit 5 (WKP) of parameter No.19696 is set to 0). Specifying G43.5 when bit 5 (WKP) of parameter No.19696 is set to 1 causes alarm PS5459.
4 Type 2 cannot be used when there is only one rotary axis or when any hypothetical axis is used. Specifying G43.5 in such a case causes alarm PS5459.
5 When using the rotary axis rollover function or the rotary axis control function, set parameter No.1260 (amount of movement per rotation of the rotary axis) to 360 degrees.

- Circular interpolation for tool center point control (type 1)

G43.4 IP_ H_ ; Starts tool center point control (type 1).



G17 : X-Y plane of the table coordinate system
G18 : Z-X plane of the table coordinate system
G19 : Y-Z plane of the table coordinate system
G02 : Clockwise (CW) circular interpolation
G03 : Counterclockwise (CCW) circular interpolation
IP : In the case of an absolute programming, the coordinate value of the end point of the tool tip movement
In the case of an incremental programming, the amount of the tool tip movement (This pertains only to two axes on the plane.)
I, J, K : Specify the distance between the start point in the rotary axis position of the block start point and the center of the arc, as seen from the programming coordinate system.
R : Arc radius R > 0: The center angle of the arc is less than 180°.
R < 0: The center angle of the arc is more than 180°.
α, β : In the case of an absolute programming, the coordinate value of the end point of the rotary axis
In the case of an incremental programming, the amount of the rotary axis movement
F : Specified speed (speed in the tangent direction of the arc as seen from the table coordinate system)
H : Tool offset number

Movement to the position specified by the G43.4 block does not constitute tool center point control. Only tool length compensation is performed. While performing interpolation for the rotary axes, the CNC controls the control points so that the tool center point moves along an arc with respect to the table (workpiece). The end of the tool center point comes to the point specified on the programming coordinate system.

⚠ CAUTION
Any command that does not move the tool center point with respect to the workpiece (one that moves the rotary axes only) must be executed in G00 or G01 mode.

- Circular interpolation for tool center point control (type 2)

G43.5 IP_ H_ Q_ ; Starts tool center point control (type 2).

G17 {**G02** / **G03**} **IP__I__J__K__ R__ F__;**

G18 {**G02** / **G03**} **IP__I__J__K__ R__ F__;**

G19 {**G02** / **G03**} **IP__I__J__K__ R__ F__;**

G17 : X-Y plane of the table coordinate system
G18 : Z-X plane of the table coordinate system
G19 : Y-Z plane of the table coordinate system
G02 : Clockwise (CW) circular interpolation
G03 : Counterclockwise (CCW) circular interpolation
IP : In the case of an absolute programming, the coordinate value of the end point of the tool tip movement
In the case of an incremental programming, the amount of the tool tip movement (This pertains only to two axes on the plane.)
I, J, K : Tool axis direction at the block end point as seen from the programming coordinate system
R : Arc radius R > 0: The center angle of the arc is less than 180°.
R < 0: The center angle of the arc is more than 180°.
F : Specified speed (speed in the tangent direction of the arc as seen from the table coordinate system)
H : Tool offset number
Q : Inclination angle of the tool (in degrees)

Movement to the position specified by the G43.5 block does not constitute tool center point control. Only tool length compensation is performed.
No rotary axes are specified. Instead, the direction of the tool end point is specified as I, J, K, as seen from the programming coordinate system (the one fixed on the table when G43.5 is specified).
In the case of a tool rotation type machine, I, J, K can be specified in the G43.5 block. But in the case of a table rotation type or composite type machine, I,J,K cannot be specified in the G43.5 block. Specifying I,J,K in the G43.5 block with a table rotation type or composite type machine causes alarm PS5421.

The significant digit number of decimals on the tool directional vector I, J, and K is up to 11 digits. However, because total digit number of integer and decimal is up to 12 digits, Maximum command digit number of integer depends on the increment system as indicated in following table.

	Increment system				
	IS-A	IS-B	IS-C	IS-D	IS-E
Maximum command digit number of integer	10	9	8	7	6

While performing interpolation for the rotary axes, the CNC controls the control points so that the tool center point moves along an arc with respect to the table (workpiece). The end of the tool center point comes to the point specified on the programming coordinate system.

⚠ CAUTION
1 Only arc radius R can be specified. (The distance from the start point to the center of the arc cannot be specified using I, J, and K.)
2 A round circle (the start point and end point are the same) cannot be specified. Any command that does not move the tool center point with respect to the workpiece (one that moves the rotary axes only) must be executed in G00 or G01 mode.
3 See the CAUTION box for tool center point control (type 2).

- Helical interpolation for tool center point control (type 1)

G43.4 IP_ H ; Starts tool center point control (type 1).

G17 {**G02**, **G03**} **IP__** {**I__J__K__**, **R__**} **α__β__γ__F__;**

G18 {**G02**, **G03**} **IP__** {**I__J__K__**, **R__**} **α__β__γ__F__;**

G19 {**G02**, **G03**} **IP__** {**I__J__K__**, **R__**} **α__β__γ__F__;**

G17 : X-Y plane of the table coordinate system
G18 : Z-X plane of the table coordinate system
G19 : Y-Z plane of the table coordinate system
G02 : Clockwise (CW) circular interpolation
G03 : Counterclockwise (CCW) circular interpolation
IP : In the case of an absolute programming, the coordinate value of the end point of the tool tip movement
In the case of an incremental programming, the amount of the tool tip movement (This pertains only to two axes on the plane.)
I, J, K : Specify the distance between the start point in the rotary axis position of the block start point and the center of the arc, as seen from the programming coordinate system.
R : Arc radius R > 0: The center angle of the arc is less than 180°.
R < 0: The center angle of the arc is more than 180°.
α, β : In the case of an absolute programming, the coordinate value of the end point of the rotary axis
In the case of an incremental programming, the amount of the rotary axis movement
γ : In the case of an absolute programming, the coordinate value of the end point of the tool tip movement
In the case of an incremental programming, the amount of the tool tip movement (This pertains only to one of the linear axes subject to tool center point control which does not exist on the plane. Linear interpolation is performed simultaneously with circular interpolation.)
F : Specified speed (speed in the tangent direction of the arc as seen from the table coordinate system)
H : Tool offset number

Movement to the position specified by the G43.4 block does not constitute tool center point control. Only tool length compensation is performed.

Because the specified speed is usually the speed in the tangent direction of the arc, the speed of the linear axis, when seen from the table coordinate system, is: $F \times \frac{\text{Length of the linear axis}}{\text{Length of the arc}}$.

Depending on bit 5 (HTG) of parameter No.1403, the specified speed varies as shown in the following table.

Bit 5 (HTG) of parameter No.1403	
0	**1**
Tangential speed of the arc	Synthetic speed of the linear axis speed and tangential speed

While performing interpolation for the rotary axes, the CNC controls the control points so that the tool center point moves helically with respect to the table (workpiece). The end of the tool center point comes to the point specified on the programming coordinate system.

⚠ CAUTION
Any command that does not move the tool center point with respect to the workpiece (one that moves the rotary axes only) must be executed in G00 or G01 mode.

- Helical interpolation for tool center point control (type 2)

G43.5 IP_ H_ Q_; Starts tool center point control (type 2).

G17 {**G02** / **G03**} **IP__I__J__K__ R__γ__F__;**

G18 {**G02** / **G03**} **IP__I__J__K__ R__γ__F__;**

G19 {**G02** / **G03**} **IP__I__J__K__ R__γ__F__;**

G17 : X-Y plane of the table coordinate system
G18 : Z-X plane of the table coordinate system
G19 : Y-Z plane of the table coordinate system
G02 : Clockwise (CW) circular interpolation
G03 : Counterclockwise (CCW) circular interpolation
IP : In the case of an absolute programming, the coordinate value of the end point of the tool tip movement
In the case of an incremental programming, the amount of the tool tip movement (This pertains only to two axes on the plane.)
I, J, K : Tool axis direction at the block end point as seen from the programming coordinate system
R : Arc radius R > 0: The center angle of the arc is less than 180°.
R < 0: The center angle of the arc is more than 180°.
γ : In the case of an absolute programming, the coordinate value of the end point of the tool tip movement
In the case of an incremental programming, the amount of the tool tip movement (This pertains only to one of the linear axes subject to tool center point control which does not exist on the plane. Linear interpolation is performed simultaneously with circular interpolation.)
F : Specified speed (speed in the tangent direction of the arc as seen from the table coordinate system)
H : Tool offset number
Q : Inclination angle of the tool (in degrees)

Movement to the position specified by the G43.5 block does not constitute tool center point control. Only tool length compensation is performed.
Because the specified speed is the speed in the tangent direction of the arc, the speed of the linear axis, when seen from the table coordinate system, is: $F \times \frac{\text{Length of the linear axis}}{\text{Length of the arc}}$.

Depending on bit 5 (HTG) of parameter No.1403, the specified speed varies as shown in the following table.

Bit 5 (HTG) of parameter No.1403	
0	**1**
Tangential speed of the arc	Synthetic speed of the linear axis speed and tangential speed

No rotary axes are specified. Instead, the direction of the tool end point is specified as I, J, K, as seen from the programming coordinate system (the one fixed on the table when G43.5 is specified).

In the case of a tool rotation type machine, I, J, K can be specified in the G43.5 block. But in the case of a table rotation type or composite type machine, I,J,K cannot be specified in the G43.5 block. Specifying I,J,K in the G43.5 block with a table rotation type or composite type machine causes alarm PS5421.

The significant digit number of decimals on the tool directional vector I, J, and K is up to 11 digits. However, because total digit number of integer and decimal is up to 12 digits, Maximum command digit number of integer depends on the increment system as indicated in following table.

	Increment system				
	IS-A	IS-B	IS-C	IS-D	IS-E
Maximum command digit number of integer	10	9	8	7	6

While performing interpolation for the rotary axes, the CNC controls the control points so that the tool center point moves helically with respect to the table (workpiece). The end of the tool center point comes to the point specified on the programming coordinate system.

⚠ CAUTION
1 Only arc radius R can be specified. (The distance from the start point to the center of the arc cannot be specified using I, J, and K.)
2 A round circle cannot be specified.
3 Any command that does not move the tool center point with respect to the workpiece (one that moves the rotary axes only) must be executed in G00 or G01 mode.
4 See the CAUTION box for tool center point control (type 2).

- Tool center point control cancellation command

G49 IP_ α_ β_ ; Cancels tool center point control.

IP : In the case of an absolute programming, the coordinate value of the end point of the tool control point movement
In the case of an incremental programming, the amount of the tool control point movement

α, β : In the case of an absolute programming, the coordinate value of the end point of the rotary axes
In the case of an incremental programming, the amount of the rotary axis movement

The cancellation block for tool center point control is the one that controls buffering.

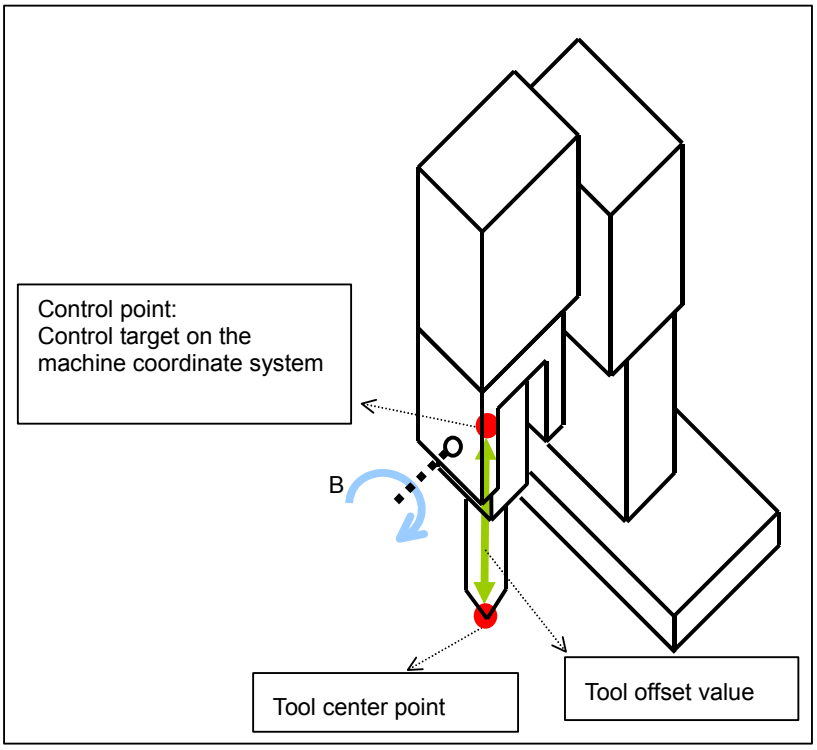


Fig. 22.1 (e) Control point and tool center point

> ⚠ **CAUTION**
> The G49 command must be executed in G00 or G01 mode.

- Inclination angle of the tool

In the case of tool center point control of type 2, the inclination angle of the tool can be specified using address Q of G43.5. The inclination angle of the tool represents how inclined the tool direction is toward the proceeding direction with respect to the direction specified by (I, J, K) at the time of machining on the plane produced by the tool direction specified by (I, J, K) and the proceeding direction on the programming coordinate system. (See the Fig. 22.1 (f).)

Overviewly, the normal direction on the machining plane is specified by (I, J, K). If the tool direction needs to be inclined toward the proceeding direction with respect to the normal direction at the time of machining, perform compensation using Q.

If the direction specified by (I, J, K) matches the direction in which the tool moves at the time of machining, commanding Q is unnecessary.

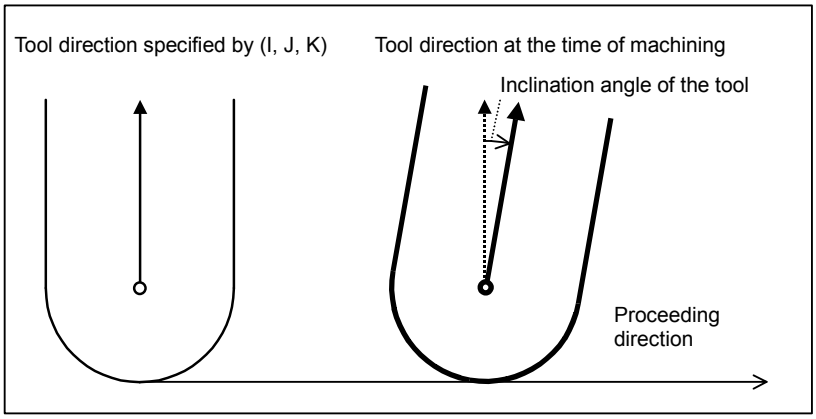


Fig. 22.1 (f)

Example)

To incline the tool by two degrees toward the proceeding direction at the time of machining, enter the following command:

G43.5 I_ J_ K_ H_ Q2.0

Explanation

- When a coordinate system fixed on the table is used as the programming coordinate system

The programming coordinate system is the coordinate system in which a program is executed with tool center point control.
If bit 5 (WKP) of parameter No. 19696 is set to '0', the workpiece coordinate system fixed on the table is the programming coordinate system. This coordinate system rotates with the rotation of the table. Because it is fixed on the table, the coordinate system is called the table coordinate system.
It does not rotate with the rotation of the tool head.

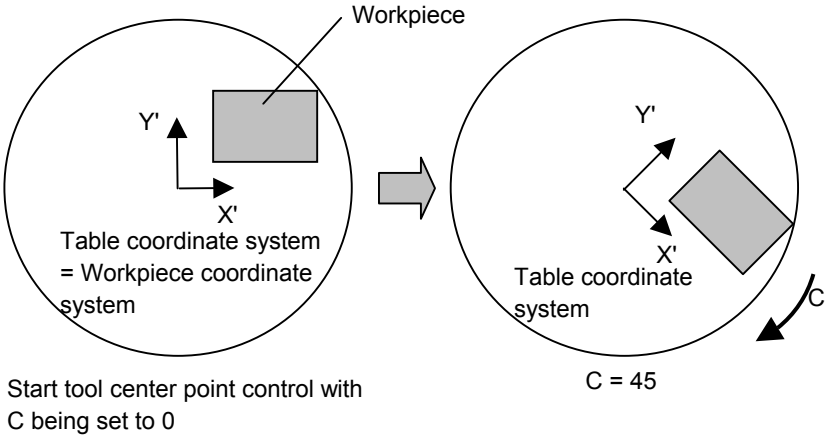
The X, Y, and Z commands are assumed to be specified in the table coordinate system.
For type 2, the tool direction as seen from the coordinate system fixed on the table is specified with I, J, and K.

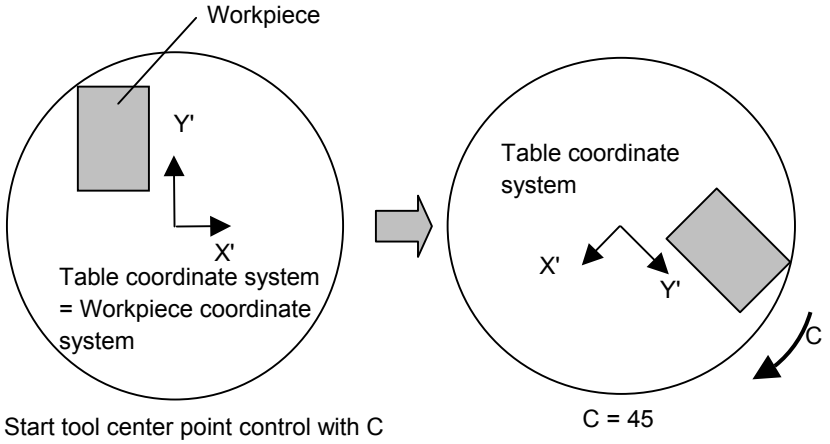
When the X, Y, or Z command is executed, the tool center performs linear movement (in G01 or G00 mode) or circular movement (in G02 or G03 mode) in relation to the table (object to be machined).
As for feedrate, feedrate control is performed so that the feedrate of the tool center in relation to the table (object to be machined) becomes the specified feedrate.

The time at which the workpiece coordinate system is fixed on the table differs depending on the setting of bit 5 (INZ) of parameter No. 19754.

If bit 5 (INZ) of parameter No. 19754 is 0, the workpiece coordinate system is fixed on the table in the state in which tool center point control is started, and subsequently, the coordinate system becomes a table coordinate system that rotates with the rotation of the table. If there is movement of a table rotation axis in the block in which tool center point control is started, the workpiece coordinate system is fixed on the table after the end of table movement.

Example of operation performed if bit 5 (INZ) of parameter No. 19754 is 0:
Assume that the table rotation axis rotating about the Z-axis is the C-axis.

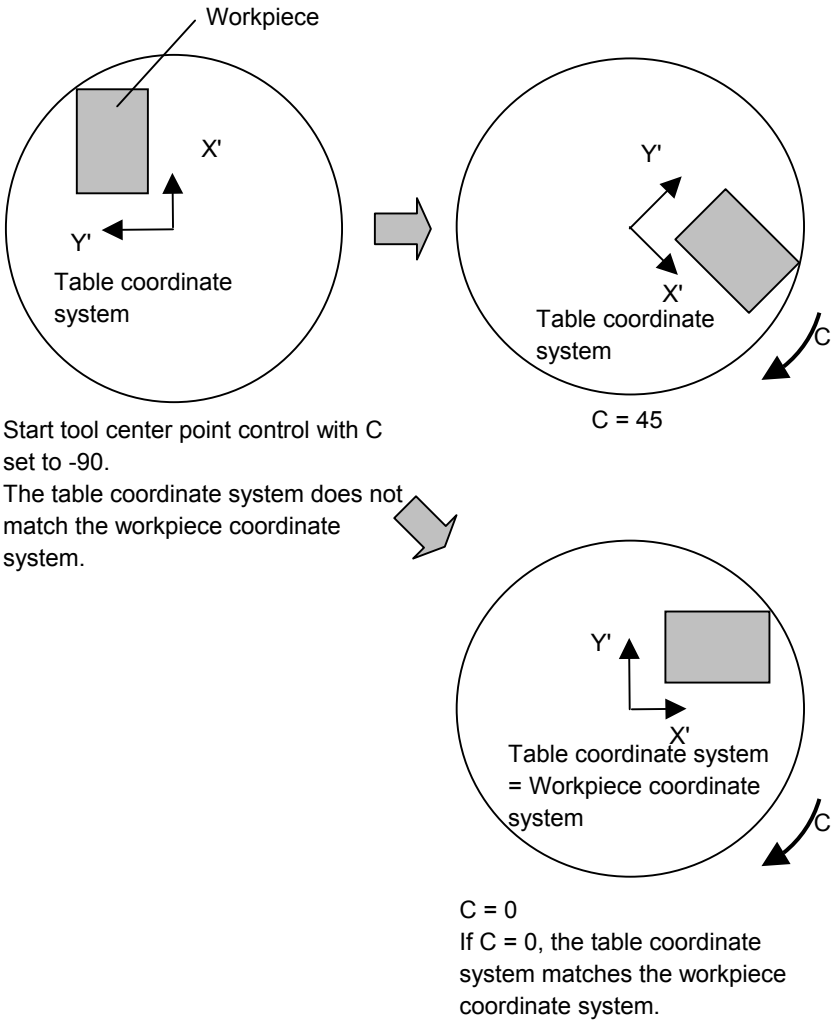




If bit 5 (INZ) of parameter No. 19754 is 0, and the program has been created by assuming that the workpiece coordinate system is fixed on the table in the state in which the position of the table rotation axis is 0, the position of the table rotation axis must be set to 0 at the start of tool center point control.
If tool center point control is turned on and off a number of times, the position of the table rotation axis must be moved to 0 each time it is turned on.

If bit 5 (INZ) of parameter No. 19754 is 1, the workpiece coordinate system is fixed on the table in the state in which the position of the table rotation axis is 0, regardless of the position of the table rotation axis at the start of tool center point control.

Example of operation performed if bit 5 (INZ) of parameter No. 19754 is 1:
Assume that the table rotation axis rotating about the Z-axis is the C-axis.



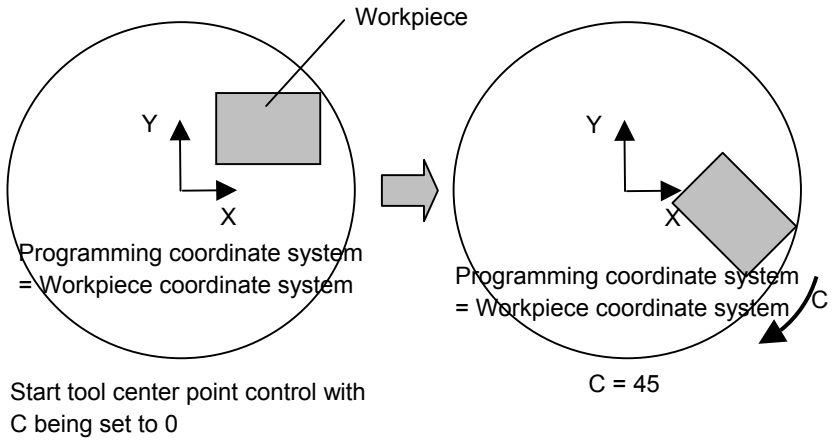
If bit 5 (INZ) of parameter No. 19754 is 1, and tool center point control is started in the state in which the position of the table rotation axis is not 0, the table coordinate system is different from the workpiece coordinate system in the block in which it is started.
If there is an axis movement command in the block in which it is started, the movement is performed on the workpiece coordinate system.
The movements in the subsequent blocks will be movements in the table coordinate system.

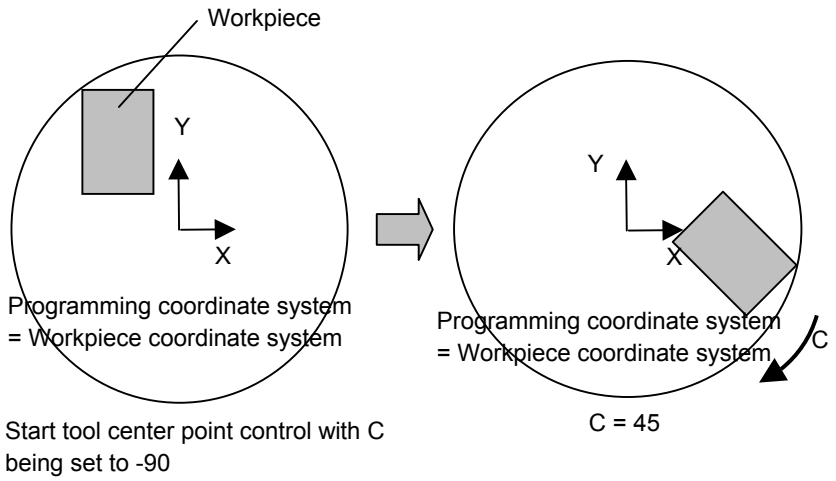
- When the workpiece coordinate system is used as the programming coordinate system

If bit 5 (WKP) of parameter No. 19696 is 1, the workpiece coordinate system is the programming coordinate system.
In this case, the programming coordinate system does not rotate with the rotation of the table, but remains fixed on the workpiece coordinate system.
For X, Y, and Z, the end point position as seen from the workpiece coordinate system after the rotation of the table is specified.
Type 2 cannot be used. If G43.5 is specified in the state in which bit 5 (WKP) of parameter No. 19696 is 1, alarm PS5459 is issued.

As with the case in which bit 5 (WKP) of parameter No. 19696 is 0, when the X, Y, or Z command is executed, the tool center performs linear movement (in G01 or G00 mode) or circular movement (in G02 or G03 mode) in relation to the table (object to be machined).
Similarly, as for feedrate, feedrate control is performed so that the feedrate of the tool center in relation to the table (object to be machined) becomes the specified feedrate.

Example of operation performed if bit 5 (WKP) of parameter No. 19696 is 1:





- Notes on performing circular interpolation and helical interpolation when using the workpiece coordinate system as the programming coordinate system

- The start point, end point, and center of an arc change as the table rotation axis rotates.

- I, J, K commands the vector of the block start point to the center of the arc from the start point in the rotary axis position.
- Note the following:
  <1> Only a table rotation axis normal to a selected plane can be rotated during circular interpolation.
  <2> During circular interpolation, those table rotation axes not normal to a selected plane must continue to be at the same position as when tool center point control is started.
  If <1> or <2> is not satisfied, the alarm PS5421 is issued.
  No restriction is imposed on rotation on a tool rotation axis.

- Program examples

- In the case of a composite type machine
  Descriptions are based on the following machine configuration (Fig. 22.1 (g)).

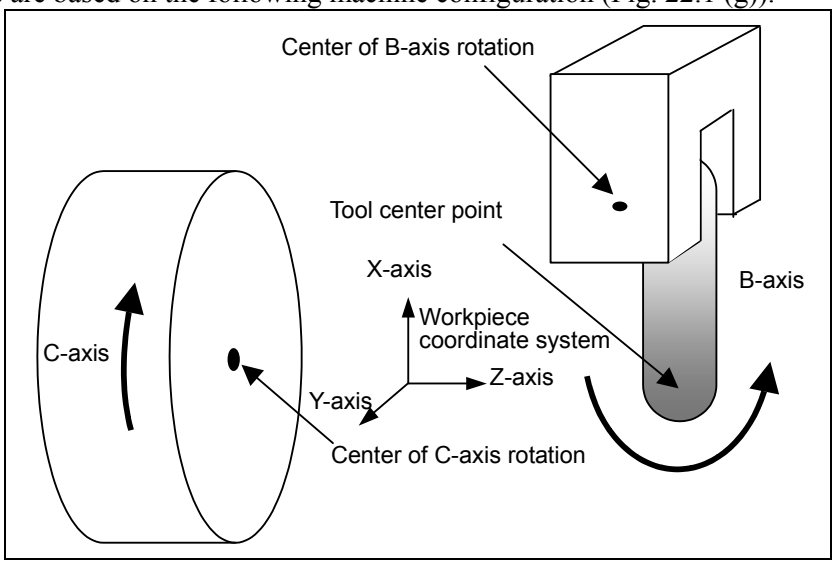


Fig. 22.1 (g)

When the G17 (X-Y plane) command is executed

After the G43.4 command, the X-Y plane is selected using the G17 command and circular interpolation is performed by rotating the C-axis (table rotation axis) (including those cases where the C-axis moves before the G43.4 command). → This case corresponds to <1> and allows circular interpolation.

Example)

```
 :
(G01 C90. ;)
G43.4 H1 ;
G17 G02 IP IR   B10. C20. ;
 :
```

IP: Coordinates of the end point
IR: Arc radius

When the G18 (Z-X plane) or G19 (Y-Z plane) command is executed

After the G43.4 command, the Z-X plane is selected using the G18 command and circular interpolation is performed without rotating the C-axis (including those cases where the C-axis moves before the G43.4 command). → This case corresponds to <2> and allows circular interpolation.

The same is also true when the G19 command is used.

Example)

```
 :
(G01 C90. ;)
G43.4 H1 ;
G18 G02 IP IR ;
```

:

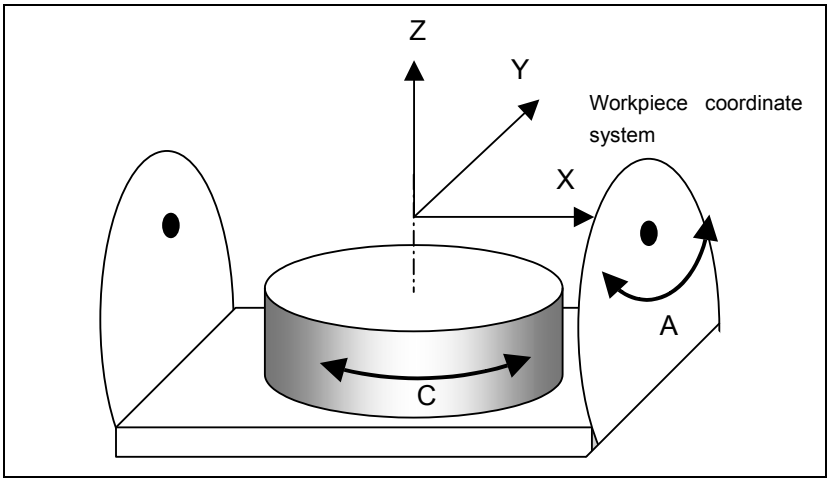
After the G43.4 command, the Z-X plane is selected using the G18 command and the C-axis is rotated during circular interpolation . → Alarm (violation of <2>)
The same is also true when the G19 command is used.
Example)

```
:
G43.4 H1 ;
G18 G02 IP IR C20. ;
:
```

After the G43.4 command, the Z-X plane is selected using the G18 command and circular interpolation is performed after rotating the C-axis. → Alarm (violation of <2>)
The same is also true when the G19 command is used.
Example)

```
:
G43.4 H1 ;
G01 C10.
G18 G02 IP IR ;
:
```

- In the case of a table rotation type machine
  Descriptions are based on the following machine configuration.
  A table rotation type machine can be considered equivalent to a composite type machine if any of its two table rotation axes does not move.



When the G17 (X-Y plane) command is executed
The master axis (A-axis) moves before the G43.4 command and, after the G43.4 command, circular interpolation is performed without rotating the A- or C-axis (including those cases where the C-axis moves before the G43.4 command). → This case corresponds to <2> and allows circular interpolation.
Example)

```
:
G01 A90. (C10.);
G43.4 H1 ;
G17 G02 IP IR ;
:
```

The master axis (A-axis) moves before the G43.4 command and, after the G43.4 command, circular interpolation is performed using the G17 (X-Y plane) command by rotating the C-axis, or the C-axis is rotated during circular interpolation. → Alarm (violation of <2>)

Example)

```
  :
G01 A90.;
G43.4 H1 ;
G01 C10. ;
G17 G02 IP IR ;
  :
  :
G01 A90.;
G43.4 H1 ;
G17 G02 IP IR C10.;
  :
```

After the G43.4 command, the A-axis is moved and circular interpolation is performed using the G17 command (X-Y plane). → Alarm (violation of <2>)

Example)

```
  :
G43.4 H1 ;
G01 A10. ;
G17 G02 IP IR C10.;
  :
```

G19 (Y-Z plane) command is executed

The G43.4 command is executed after moving the A-axis and circular interpolation is performed while rotating the A-axis using the G19 (Y-Z plane) command. → This case corresponds to <1> and allows circular interpolation.

Example)

```
  :
G01 A90. ;
G43.4 H1 ;
G19 G02 IP IR A10. ;
  :
```

After the G43.4 command, the C-axis is rotated and circular interpolation is performed using the G19 (Y-Z plane) command. → Alarm (violation of <2>)

Example)

```
  :
G43.4 H1 ;
G01 C10. ;
G19 G02 IP IR ;
  :
```

When the G18 (Z-X plane) command is executed

The G43.4 command is executed after moving the A- and C-axes, and circular interpolation is performed using the G18 (Z-X plane) command without moving any rotary axis. → This case corresponds to <2> and allows circular interpolation.

Example)

```
  :
G01 A90. C10. ;
G43.4 H1 ;
G18 G02 IP IR;
  :
```

After the G43.4 command, circular interpolation is performed using the G18 command (Z-X plane) by moving any of the rotary axes. → Alarm (violation of <2>)

Example)

```
  :
G43.4 H1 ;
G01 A10. (C10.)
G18 G02 IP IR;
  :
```

- Tool center point control command

During tool center point control, the command specifies the location of each block end point as seen from the programming coordinate system.
The program specifies the tool center point.
As for the rotary axis, the command specifies the coordinate values of each block end point in the case of type 1 or the tool direction at each block end point in the case of type 2.
The feedrate is specified by the tangential speed relative to the workpiece (the tool's relative speed as opposed to the workpiece), which is represented by F.

- Commands that can be specified during tool center point control

The commands that can be specified during tool center point control are linear interpolation (G01), positioning (G00), circular interpolation (G02, G03), and helical interpolation (G02, G03).
When linear interpolation (G01) is specified during tool center point control, speed control is exerted so that the tool center point moves at the specified speed.
The circular interpolation command (G02, G03) controls the tangential speed of the arc path along which the tool center point moves.
The helical interpolation command (G02, G03) controls the tangential speed of the arc path along which the tool center point moves or a synthetic speed including that of the helical axis. (This is dependent on the setting of bit 5 (HTG) of parameter No.1403.)

NOTE

When the tool center point control mode is entered, acceleration/deceleration before look ahead interpolation is enabled automatically. Be sure to specify the following parameters:

(1) Bit 1 (LRP) of parameter No.1401=1 :
Linear rapid traverse
(2) Bit 5 (FRP) of parameter No.19501=1 :
Acceleration/deceleration before interpolation is used for rapid traverse.
(3) Parameter No.1671 :
Acceleration before interpolation for rapid traverse.
(4) Parameter No.1672 :
Change time for bell-shaped acceleration before interpolation for rapid traverse.
(5) Parameter No.1660 :
Maximum permissible acceleration for acceleration/deceleration before interpolation
(6) Parameter No.1432 :
Maximum cutting feedrate for all axes in the acceleration/deceleration before interpolation

If they are not specified, alarm PS5420 or PS5011 is generated.

- Rotary axis command

If a command is specified during tool center point control that prohibits the tool center point from moving with respect to the workpiece, the maximum cutting speed (parameter No.1432) is assumed as the feedrate of the rotary axis when bit 6 (RFC) of parameter No.19696 is 0, and the speed specified by F is assumed when bit 6 (RFC) of parameter No.19696 is 1.
The rotary axis command cannot be specified during tool center point control of type 2. Specifying the command with type 2 causes alarm PS5421.

- The moving distance of the rotary axis is long compared to that of the linear axis

If the moving distance of the rotary axis is long compared to that of the linear axis, the rotary axis moves faster so that the tool center point moves at the specified speed, possibly resulting in the tool center point traveling on an inadequate path.
In such a case, it is possible to exert control to slow down the speed and ensure that the tool center point travels on the specified path, by setting bit 6 (CRS) of parameter No.19746 to 1. When using this control, specify in parameter No.19751 (for rapid traverse) and parameter No.19752 (for cutting feed) the extent of deviation of the path at which the speed is to be slowed down (maximum allowable deviation of the path). When 0 is specified, the least input increment is regarded as the maximum allowable deviation of the path.

- Tool behavior at startup and cancellation

When tool center point control is started (G43.4/G43.5) or canceled (G49), the tool moves by a tool offset value.
Compensation vector calculation is performed only at the end of a block.

- Setting for workpiece coordinate system in tool center point control

In tool rotation type, G92 (In case of G code system A in Lathe system, G code is G50.) can be commanded. Therefore, position of tool center point on the workpiece coordinate system can be freely changed.

Example)

Tool length offset value of H1
200.0

```
O0001
N01G43.4H1
N02G90G00X100.0Y100.0Z100.0B45.0C0
N03G92X0Y0Z0
N04X200.0Y0Z0
```

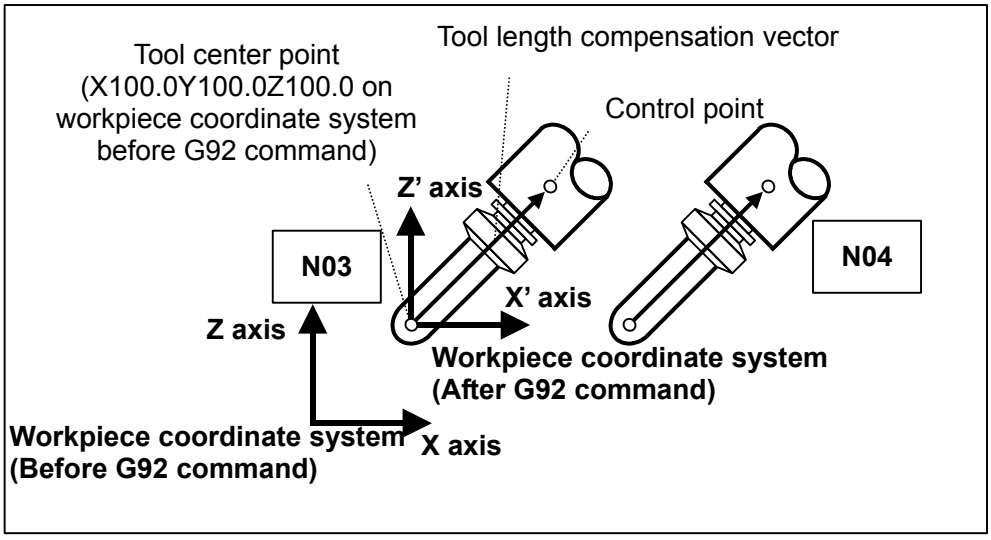


Fig.22.1 (h) G92 command in tool center point control

If N03 block on sample program O0001 is executed, tool center point position on workpiece coordinate system becomes commanded position at G92 block. (Fig.22.1 (h))

- Workpiece coordinate system preset in tool center point control

In tool rotation type, G92.1 (In case of G code system A in Lathe system, G code is G50.3.) can be commanded. Therefore, a workpiece coordinate system shifted by manual intervention or G92 command can be preset to the pre-shift workpiece coordinate system. The latter system is displaced from the machine zero point by a workpiece origin offset value.

Example)

Tool length offset value of H1
200.0

Workpiece origin offset value of G54
X200.0 Y200.0 Z200.0

```
O0002
N01G54
N02G43.4H1
N03G90G00X100.0Y100.0Z100.0B45.0C0
N04G92X0Y0Z0
N05G92.1X0Y0Z0
N06X200.0Y0Z0
```

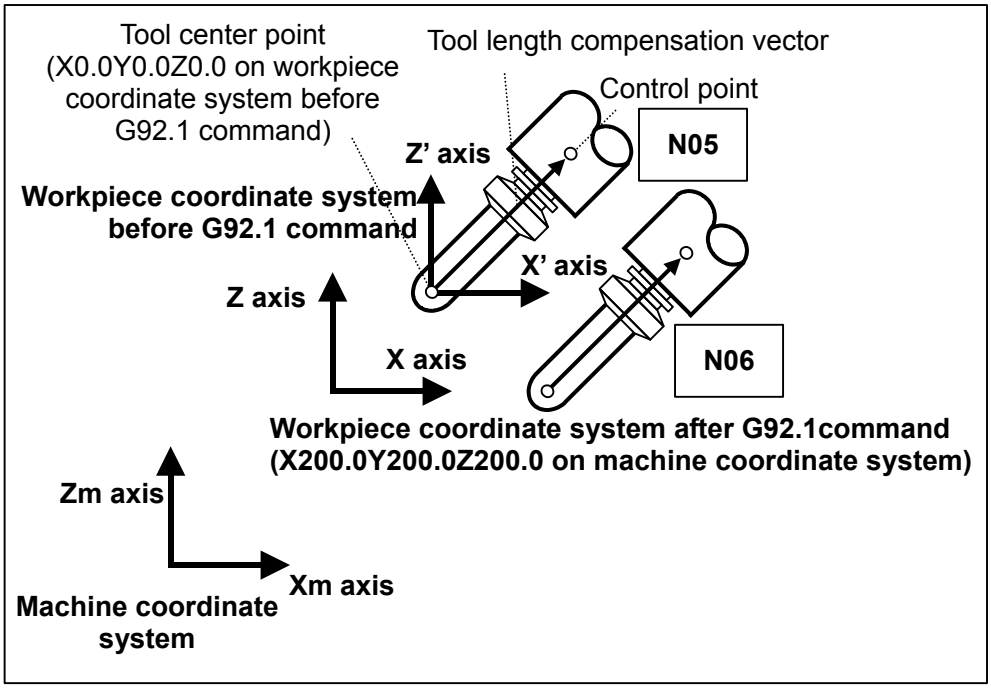


Fig.22.1 (i) G92.1 command in tool center point control

If N05 block on sample program O0002 is executed, the workpiece coordinate system is displaced from the machine zero point by a workpiece origin offset value. (Fig.22.1 (i))

- Current position display during tool center point control

During tool center point control, the position of the control point (rotation center of the tool rotation axis) is displayed as the machine coordinate.
When bit 5 (WKP) of parameter No.19696 is 0, whether to use absolute or relative coordinates can be selected using bit 2 (DET) of parameter No.19608.
If bit 2 (DET) of parameter No.19608 is 0, the position of the tool center point on the programming coordinate system is displayed.
If bit 2 (DET) of parameter No.19608 is 1, the position of the tool center point in the workpiece coordinate system is displayed.

- Actual cutting feedrate display during tool center point control

By using bit 6 (TFD) of parameter No. 11260, the actual cutting feedrate display during tool center point control can be changed as described below.

- When TFD = 0:
  The actual feedrate at the control point is displayed.
- When TFD = 1:
  The actual feedrate at the tool center point is displayed.
  Note that the correct feedrate may not be displayed in the event of manual intervention or an interrupt with the manual handle.
  The actual feedrate at the control point is displayed during synchronous, composite, or superimposed control and during workpiece setting error compensation.

- Distance to go display during tool center point control

By using bit 4 (TDG) of parameter No. 11260, the display of the distance to go during tool center point control can be changed as described below.

- When TDG = 0:
  The distance to go in the table coordinate system is displayed.
  Even if TDG = 0, the distance to go in the machine coordinate system is displayed in the following modes:
  - 3-dimensional coordinate system conversion
  - Tilted working plane indexing command
  - Workpiece setting error compensation
- When TDG = 1:
  The distance to go in the machine coordinate system is displayed.

- Inverse time feed

In inverse time feed during tool center point control, if linear interpolation (G01) is performed, the inverse time to be specified with an F code (FRN) is:

$$FRN = \frac{1}{\text{Time (min)}} = \frac{\text{Movement speed of the tool center point}}{\text{Movement distance of the tool center point}}$$

If the tool center point does not move in relation to the workpiece, but movement is made along the rotation axis, the "movement distance of the tool center point" is the amount of travel along the rotation axis.
In a tool rotation type machine, to perform inverse time feed during tool center point control, bit 7 (TIT) of parameter No. 19605 must be set to 0.
While inverse time feed is being specified during tool center point control, manual intervention cannot be performed. Do not perform manual intervention.

- Feed per revolution

In a tool rotation type machine, to perform feed per revolution during center point control, bit 7 (TIT) of parameter No. 19605 must be set to 0.
While feed per revolution is being specified during tool center point control, manual intervention cannot be performed. Do not perform manual intervention.

- External deceleration during tool center point control

During tool center point control, external deceleration is applied to the feedrate not at the tool center point, but at the control point.

- Tool offset

If tool offsets are used based on tool numbers, tool center point control is carried out using the tool length compensation value corresponding to the relevant tool number (T code).
If tool life management is used, tool center point control is carried out using the tool length compensation value corresponding to the tool in use.

- Function for placing priority on the path rather than on speed

If, in a block specifying a movement on a rotary axis, a high speed (speed command or rapid traverse rate) is specified or the travel distance on the rotary axis is larger than the travel distance of the tool center point, an excessive path deviation can occur in the middle of the block when a movement is made by the specified speed.
In such a case, priority can be placed on the path instead of the speed so that the speed can be automatically lowered to prevent a deviation from the path.

This function is enabled by setting bit 6 (CRS) of parameter No. 19746 to 1.
Set a maximum allowable path deviation in parameter No. 19751 (for rapid traverse) or in parameter No. 19751 (for cutting feed).
When 0 is set as a maximum allowable path deviation, the specification of the minimum unit of setting data of the parameter is assumed.
When a negative value is set, the function for placing priority on the path is disabled.

When a judgment is made to lower the speed, the speed of the entire block is lowered.
As a result of speed reduction, the path error may be smaller than a parameter-set value due to a calculation error.

- Canned cycle for drilling during tool center point control

When bit 5 (TFA) of parameter No. 5105 is set to 1, if a canned cycle for drilling being subjected to tool center point control is performed when the position of the rotation axis that determines the tool axis direction is not ±90° × n (n=0, 1, 2...) on the workpiece coordinate system, alarm PS5424 (invalid tool axis direction) occurs.

> ⚠ **CAUTION**
> A canned cycle for drilling being subjected to tool center point control operates in the direction of the program coordinate system. It does not always operates in the current tool axis direction. Be extremely careful about programming.

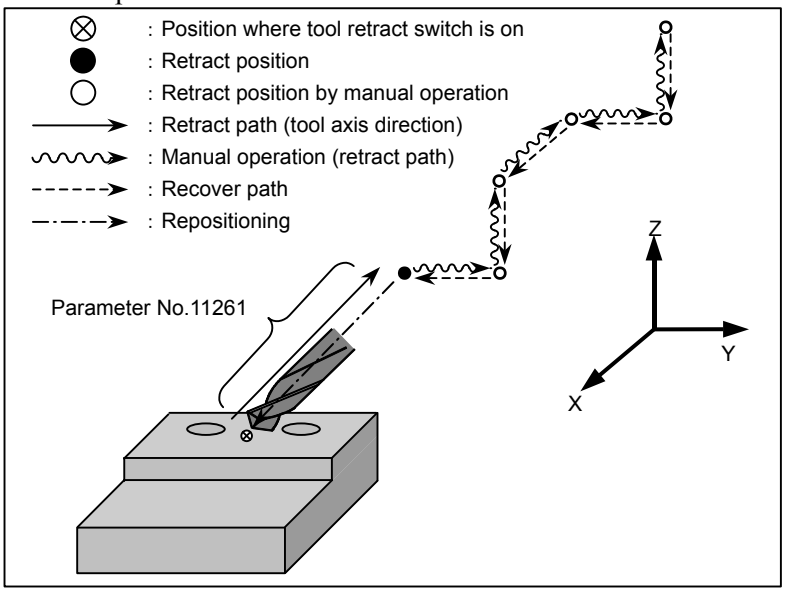
- Tool retract and recover during tool center point control

A tool retract and recover operation can be performed during tool center point control. A retract operation is performed in the tool axis direction by setting a tool axis direction retract amount in parameter No. 11261.
This function is enabled on a machine of head rotation type.
For tool retract and recover operation, refer to "TOOL RETRACT AND RECOVER" in Part for Operation.

Tool axis direction retract operation is enabled under the following conditions:

1. The G10.6 command is singly executed (no axis address is specified in the block specifying G10.6).
2. Bit 2 (RPS) of parameter No. 7040 is set to 1.
3. Tool center point control or workpiece setting error compensation are being exercised. (However, bit 0 (RCM) of parameter No. 11200 is set to 1 to perform compensation in the tool direction.)
4. A nonzero value is set in parameter No. 11261.

If the tool retract signal TRESC is set to 1 under the conditions above, a retract operation is performed in the tool axis direction. Parameter No. 11261 is used to specify a retract amount. A retract operation is performed using the value of parameter No. 11261 as an incremental retract amount.



If the tool retract signal TRESC is set to 1 during tool center point control when the tool axis direction retract conditions mentioned above are not satisfied, a retract operation of the conventional type is performed. In such a case, specify the tool center point retract position in a G10.6 block.

G10.6 alone			G10.6 X_ Y_ Z_
Bit 2 (RPS) of parameter No.7040 = 1		= 0	Conventional retract
Parameter No.11261 ≠ 0.0	= 0.0	Retract operation is not performed.	
Tool axis direction retract	Conventional retract (Parameter No. 7041)		

- Functions that automatically enable AI contour control

If tool center point control is used, it is recommended to use AI contour control.
By setting bit 2 (AAI) of parameter No. 11260 to 1, AI contour control can be automatically enabled when tool center point control is specified.
If AI contour control is enabled with this function, AI contour control is also canceled with the tool center point control cancel command (G49).

Example)

If tool center point control is specified (Bit 2 (AAI) of parameter No. 11260=1)

```
    :
G43.4H507 ;        Starts tool center point control and AI contour control mode
    :
G49 ;              Ends tool center point control and AI contour control mode
```

If AI contour control has been enabled in advance, AI contour control is not canceled with the tool center point control cancel command (G49).

Example)

If tool center point control is specified during AI contour control (Bit 2 (AAI) of parameter No. 11260=1)

```
    :
G5.1Q50;           Starts AI contour control mode
G43.4H50 ;         Starts tool center point control
    :
G49 ;              Ends tool center point control
G5.1Q0 ;           Ends AI contour control mode
```

This function is also effective to smooth TCP, tool posture control, and the cutting point command.

If bit 2 (AAI) of parameter No. 11260 is set to 1 although the AI contour control option is absent, alarm PS5420 is issued if tool center point control is specified.

Example)

If tool center point control is specified without the AI contour control option (Bit 2 (AAI) of parameter No. 11260=1)

```
    :
G43.4H50 ;         Stops operation (alarm PS5420)
    :
```

NOTE
1 If the AI contour control is enabled with the G08 command in advance, the number of blocks to be preread does not increase due to this function.
2 AI contour control is canceled with a reset.

- Disabling tool center point control during rapid traverse

Using a parameter, it is possible to select a tool path for tool center point control.

By setting bit 7 (TRC) of parameter No. 11260 to 1 during rapid traverse in tool center point control, it is possible to disable tool center point control to only the tool path, without changing the end position of the tool.

Tool rotation type:

Fig. 22.1 (j) shows a comparison of rapid traverse operations of tool center point control depending on the setting of bit 7 (TRC) of parameter No. 11260. If TRC is 1, the path of control points from the block start point to the end point is linear.

```
O0001(TOOL_TYPE)
G00 X0 Y0 Z0 B0 C0
G5.1Q1
G43.4 H1
X200.0 B90.0
```

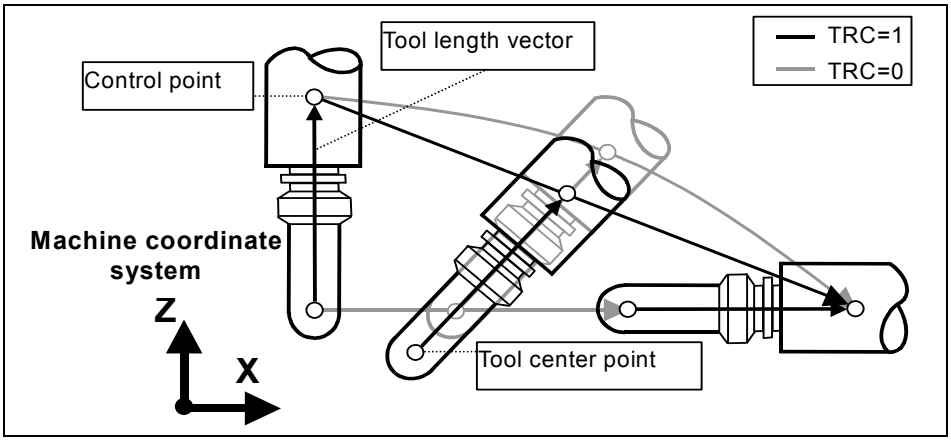


Fig. 22.1 (j) Comparison of operations depending on the setting of bit 7 (TRC) of parameter No. 11260(tool rotation type)

Table rotation type:

Fig. 22.1 (k) shows a comparison of rapid traverse operations of tool center point control depending on the setting of bit 7 (TRC) of parameter No. 11260. If TRC is 1, the path of control points from the block start point to the end point is linear.

```
O0002(TABLE_TYPE)
G00 X0 Y0 Z0 B0 C0
G5.1Q1
G43.4 H1
B90.0
```

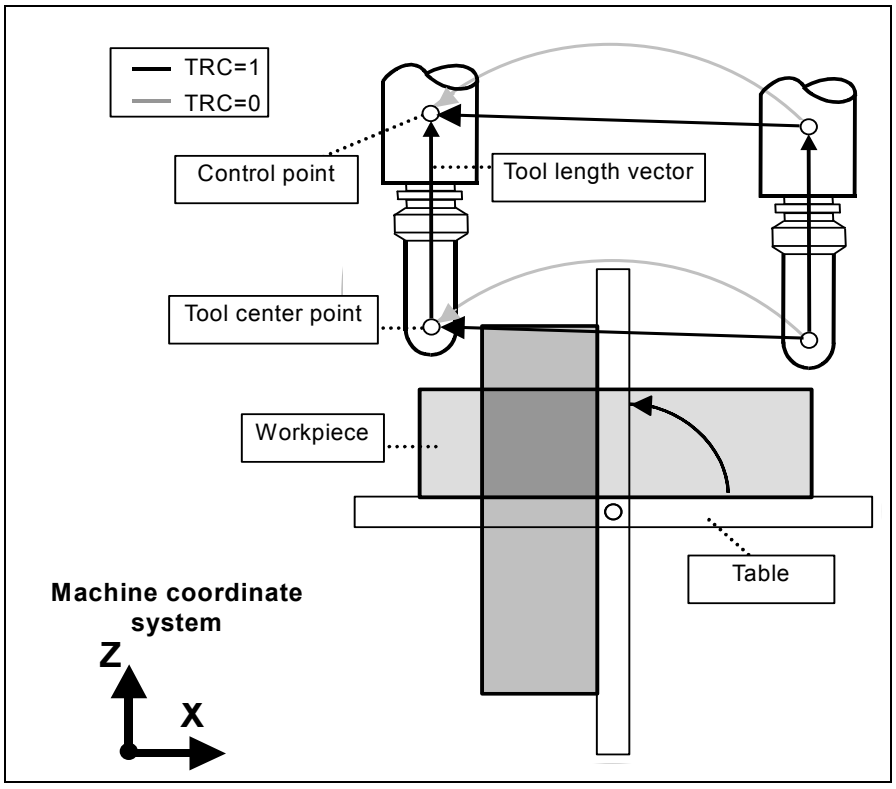


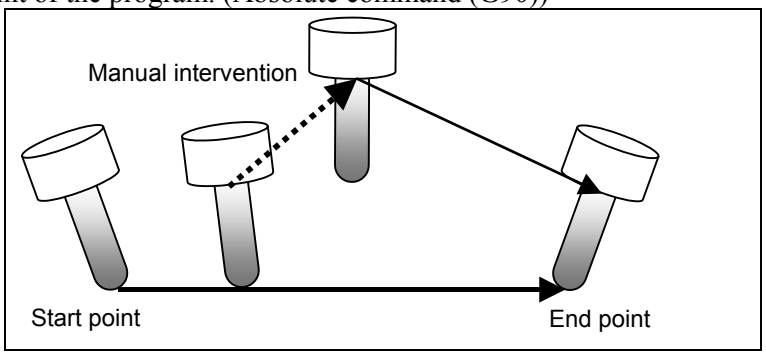
Fig. 22.1 (k) Comparison of operations depending on the setting of bit 7 (TRC) of parameter No. 11260 (table rotation type)

Manual intervention during tool center point control

- For a tool rotation type machine

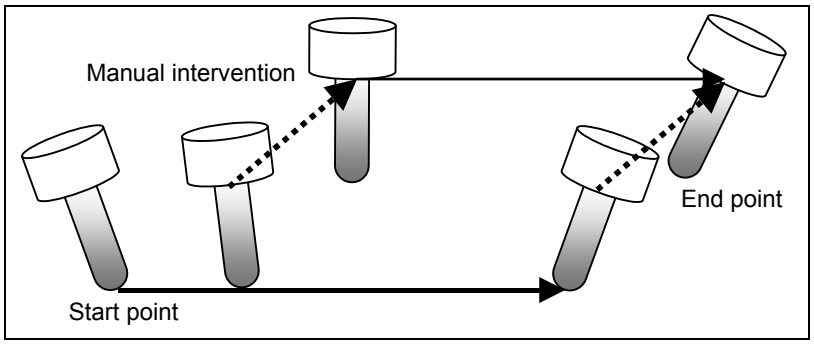
Manual absolute on (*ABSM=0)

If an operation restart is performed after manual intervention in the manual absolute on state, the tool moves to the end point of the program. (Absolute command (G90))



Manual absolute off (*ABSM=1)

If an operation restart is performed after manual intervention in manual absolute off state, the tool is positioned at the end point to which the amount of manual intervention is added, and the amount of manual intervention is also added to the subsequent operations.

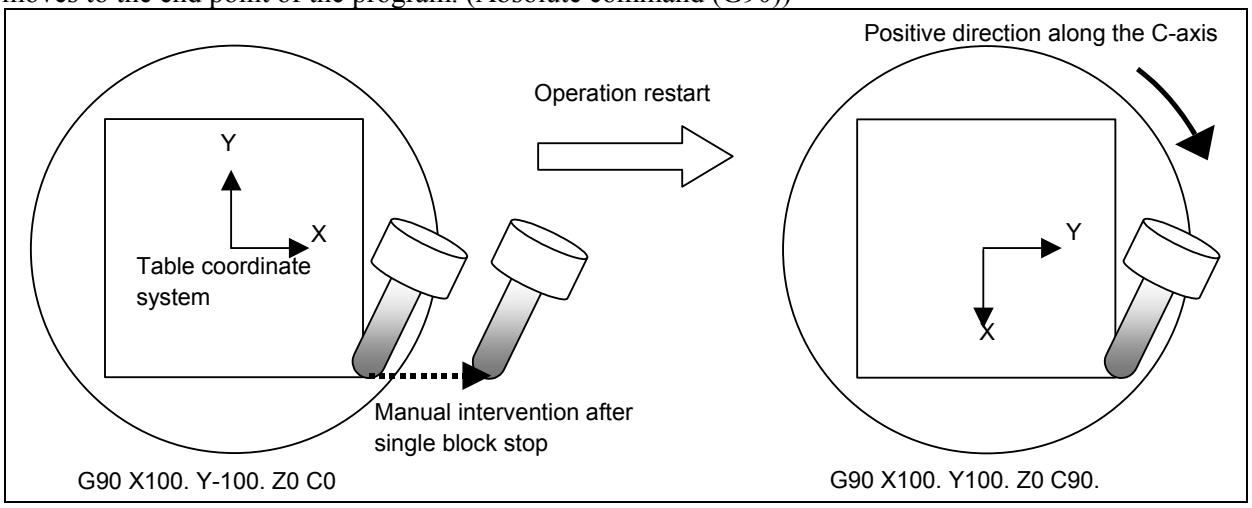


NOTE
If manual intervention is performed on a rotation axis, the center position shifts by the amount of intervention.

- For a table rotation type or composite type machine

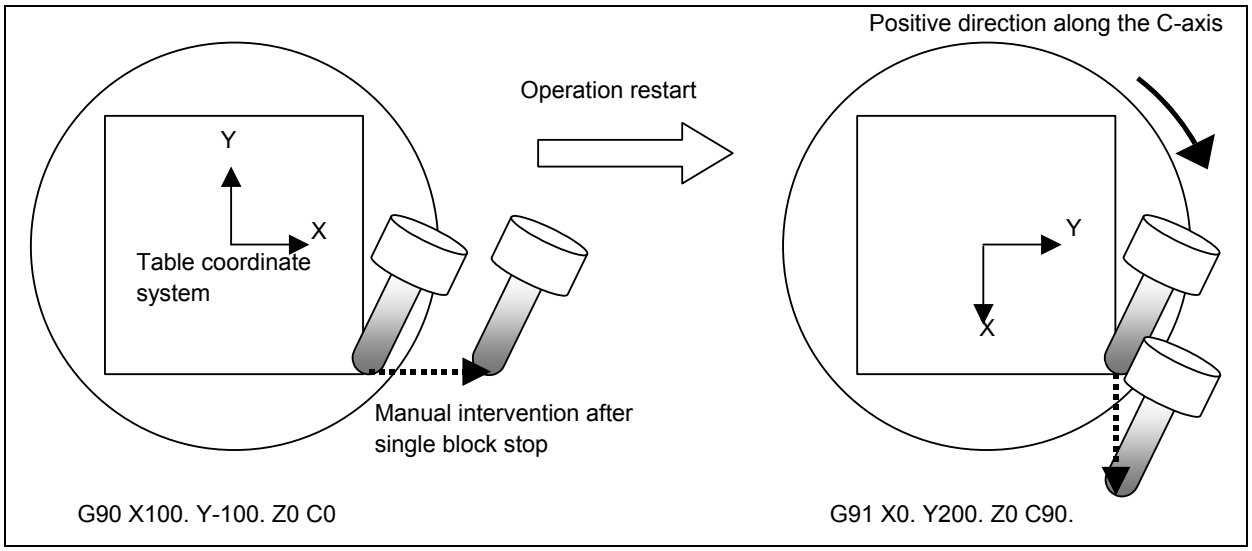
Manual absolute on (*ABSM=0)

If an operation restart is performed after manual intervention in the manual absolute on state, the tool moves to the end point of the program. (Absolute command (G90))



Manual absolute off (*ABSM=1)

If an operation restart is performed after manual intervention in the manual absolute off state, the tool moves in the state in which the amount of manual intervention is retained in the "table coordinate system".



NOTE
If manual intervention is performed in an incremental command (G91) block, the operation is the same as that performed in the manual absolute off state.
If ABS of parameter No. 7001 is 1, the operation is the same as that of an absolute command (G90).

- Angle of the rotary axis for type 2 (when the movement range is not specified)

When the direction of the tool is specified by I, J, K, Q for type 2, more than two pairs of "computed angles" of the rotary axes usually exist.
The "computed angle" is the candidate angle at which the rotary axis is to be controlled in the specified tool axis direction.
The "output angle" is determined from the "computed angle" based on the "output judgment conditions" described below.

The following descriptions assume that there is no movement range specification (parameter No.19741 - No.19744 = 0).

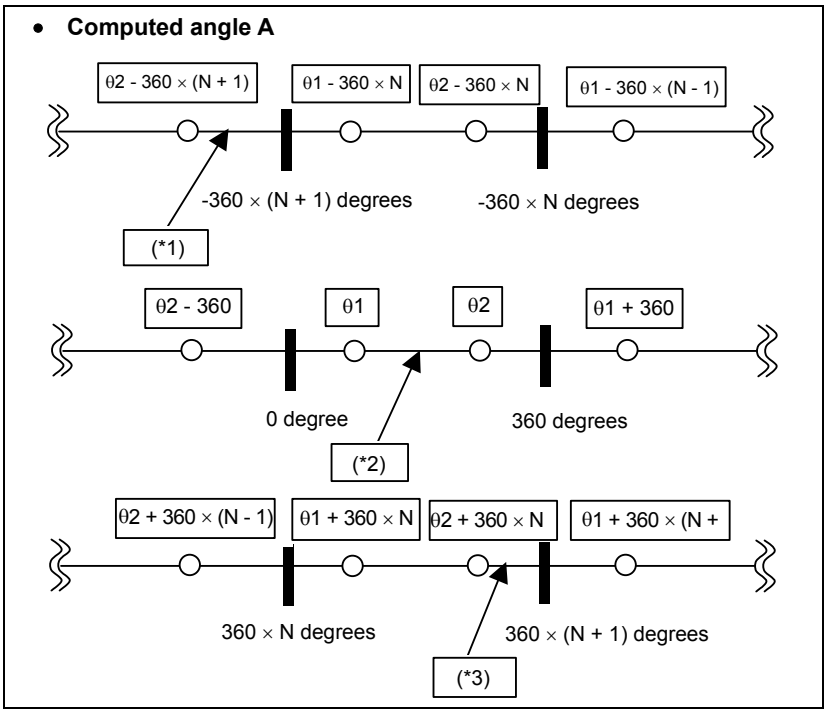
"Output judgment conditions"

Tool rotation type or table rotation type machine
<1> The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) moving angle is smaller. ↓ ↓ When the master axis moving angle is the same ↓
<2> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) moving angle is smaller. ↓ ↓ When the slave axis moving angle is the same ↓
<3> The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees). ↓ ↓ When the master axis angle is equally near to 0 degree ↓
<4> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

Composite type machine
<1> The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) moving angle is smaller. ↓ ↓ When the table moving angle is the same ↓
<2> The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) moving angle is smaller. ↓ ↓ When the tool moving angle is the same ↓
<3> The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees). ↓ ↓ When the table angle is equally near to 0 degree ↓
<4> The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

The process of judging whether the moving angle is smaller or larger as the output judgement condition is called "movement judgement."
When bit 5 (PRI) of parameter No.19608 is 1, the movement judgements for the first rotary axis and second rotary axis are made in reverse order.

The "movement judgement" process is explained below.
When the "computed angle" is within the range between 0 and 360 degrees, it is called the "basic computed angle."
Usually, two pairs of "basic computed angles" exist.
For example, assume that a tool rotation type or table rotation type machine has rotary axis A (master) and rotary axis B (slave) and that there are two pairs of basic computed angles as follows:
(A $\theta 1$ degree; B $\phi 1$ degree)
(A $\theta 2$ degrees; B $\phi 2$ degrees) where $\theta 1 \leq \theta 2$.
The "computed angle" is obtained from either of the following expressions: "basic computed angle" + 360 degrees $\times$ N or "basic computed angle" - 360 degrees $\times$ N.
The current position of rotary axis A (master) is PA, and that of rotary axis B (slave) is 0 degree.
Based on the PA angle, the "movement judgement" process is done as follows (when bit 5 (PRI) of parameter No.19608 is 0).



"Movement judgment"

When the PA angle is (*1):
The output angle is: (A θ2 - 360 × (N + 1) degrees; B ϕ2 degrees).
Namely, θ2 - 360 × (N + 1) degrees is adopted that is nearer to the computed angle of A, and ϕ2, which is the same group as θ2, is adopted as the computed angle of B.

When the PA angle is (*2):
The output angle is: (A θ1 degrees; B ϕ1 degrees).
Namely, θ1 degrees is adopted that is nearer to the computed angle of A, and ϕ1, which is the same group as θ1, is adopted as the computed angle of B.

When the PA angle is (*3):
The output angle is: (A θ + 360 × N degrees; B ϕ2 degrees).
Namely, θ2 + 360 × N degrees is adopted that is nearer to the computed angle of A, and ϕ2, which is the same group as θ2. is adopted as the computed angle of B.

When the moving angle of rotary axis A (master) is the same, a "movement judgement" is made for rotary axis B (slave) according to the "output judgment conditions."
If the "output angle" of rotary axis A is determined by the "movement judgement" for rotary axis A, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis B. Similarly, if the "output angle" of rotary axis B is determined by the "movement judgement" for rotary axis B, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis A.

The "output angle" is explained below using a tool rotation type machine as an example.
This example illustrates a machine having a "BC type tool axis Z."

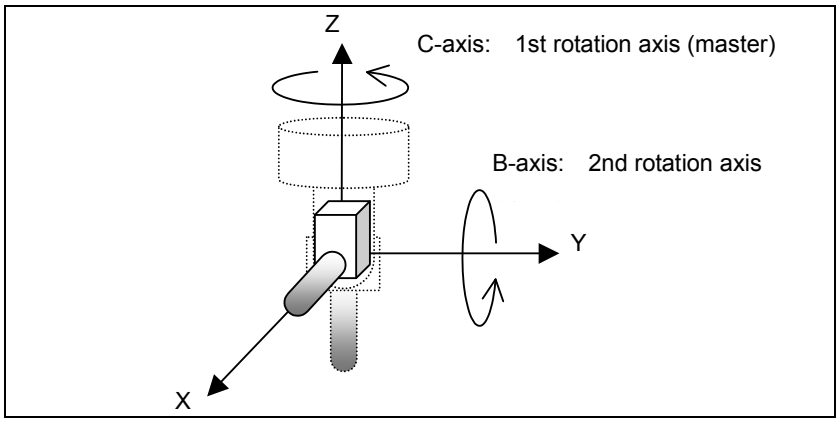


Fig. 22.1 (l) BC type tool axis Z

The following two pairs of "computed basic angles" exist that direct the tool axis toward the + X axis direction.
(B 90 degrees; C 180 degrees)
(B 270 degrees; C 0 degree)

<1> When the current rotary axis angles are (B -70 degrees; C 30 degrees)
The "output angles" are (B -90 degrees; C 0 degree).
0 degree is adopted because it is nearer to the current position (30 degrees) of the C-axis that is the master axis. For the B-axis, 270 degrees is adopted which is the same group. However, this is changed to -90 degrees (270 degrees - 360 degrees) which is the nearest to the current position of the B-axis (-70 degrees).

<2> When the current rotary axis angles are (B 80 degrees; C 500 degrees)
The "output angles" are (B 90 degrees; C 540 degrees).
540 degrees (180 degrees + 360 degrees) is adopted because it is nearer to the current position (500 degrees) of the C-axis that is the master axis. For the B-axis, 90 degrees is adopted which is the same group.

<3> When the current rotary axis angles are (B 60 degrees; C 90 degrees)
The "output angles" are (B 90 degrees; C 180 degrees).
Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. 90 degrees is adopted because it is nearer to the current position (60 degrees) of the B-axis that is the slave axis. For the C-axis, 180 degrees is adopted which is the same group.

<4> When the current rotary axis angles are (B 180 degrees; C 90 degrees)
The "output angles" are (B 270 degrees; C 0 degree).
Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. In this case, however, the two candidates are also equally near to the current position of the B-axis (180 degrees). Therefore, the candidate is adopted in which the C-axis (master axis) is nearer to 0 degree.
That is, the pair is adopted whose C axis angle is 0 degree and whose B axis angle is 270 degrees.

When the slave axis angle is 0 degree, the direction of the tool axis becomes fixed regardless of the master axis angle.
In that case, the master axis does not move from the current angle.

An explanation is shown below using a machine having a "BC type tool axis Z" as an example.

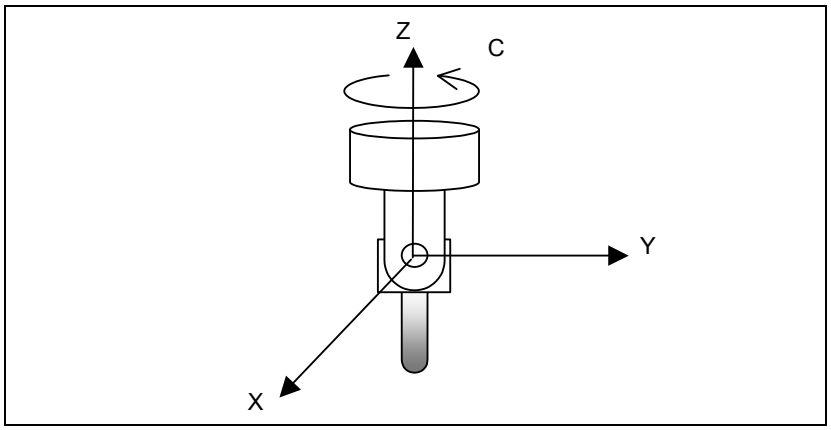


Fig. 22.1 (m) BC type tool axis Z

When the current rotary axis angles are (B 45 degrees; C 90 degrees), the "output angles" are (B 0 degree; C 90 degrees).

- Angle of the rotary axis for type 2 (when the movement range is specified)

If the upper and lower limits of the movement range of the rotary axis is specified using parameters No.19741 to No.19744, the rotary axis will move only within the specified range when the direction is specified using I, J, K, Q command for type 2 control.

Although the procedure for determining the angles is the same as that used "when the movement range is not specified," the "output angles" need to be selected from **those computed angles that are within the specified movement range for both axes**.

"Output judgment conditions"

Tool rotation type or table rotation type machine	
<1>	Of the angle pairs whose master and slave axis angles are both within the specified movement range, the rotary axis angle pair whose master axis (first rotary axis) moving angle is smaller represents the "output angles." ↓ ↓ When the master axis moving angle is the same ↓
<2>	The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) moving angle is smaller. ↓ ↓ When the slave axis moving angle is the same ↓
<3>	The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees). ↓ ↓ When the master axis angle is equally near to 0 degree ↓
<4>	The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

Composite type machine	
<1>	Of the angle pairs whose master and slave axis angles are both within the specified movement range, the rotary axis angle pair whose table (second rotary axis) moving angle is smaller represents the "output angles."
	↓
	↓ When the table moving angle is the same
	↓
<2>	The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) moving angle is smaller.
	↓
	↓ When the tool moving angle is the same
	↓
<3>	The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).
	↓
	↓ When the table angle is equally near to 0 degree
	↓
<4>	The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

When bit 5 (PRI) of parameter No.19608 is 1, the movement judgements for the first rotary axis and second rotary axis are made in reverse order.

⚠ CAUTION

1 If the lower limit of the movement range is larger than the upper limit, alarm PS5459 occurs when G43.5 is specified.
2 If no "computed angle" is found within the movement range because the range is too small, alarm PS5459 occurs.
3 If 0 is set for both parameters that specify the upper and lower limits of the movement range, the tool operates assuming that there is no range specification.
4 When the rotary axis rollover function or rotary axis control function is used (in which case, set parameter No.1260 (amount of movement per rotation of the rotary axis) to 360 degrees), the tool does not move beyond 0 degree (360 degrees) (does not take the shortcut) if the movement range is set between 0 and 360 degrees. Also, do not specify a negative value or a value larger than 360 degrees for the movement range.

An example of the "movement judgement" process is given below.
Assume that a tool rotation type or table rotation type machine has rotary axis A (master) and rotary axis B (slave) and that there are two pairs of basic computed angles as follows:
(A θ1 degree; B ϕ1 degree)
(A θ2 degrees; B ϕ2 degrees) where $\theta 1 \leq \theta 2$.
The "computed angle" is obtained from either of the following expressions: "basic computed angle" + 360 degrees × N or "basic computed angle" - 360 degrees × N.
Assume that the current positions and movement ranges of rotary axis A (master) and rotary axis B (slave) are as shown in the Fig. 22.1 (n) and Fig. 22.1 (o).

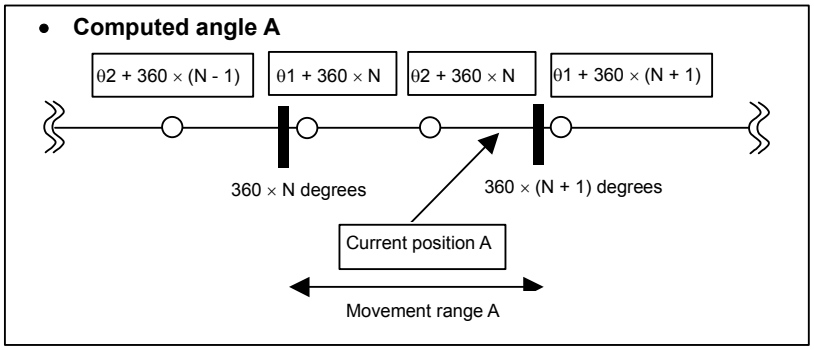


Fig. 22.1 (n) "Computed angle of rotary axis A and its current position and movement range"

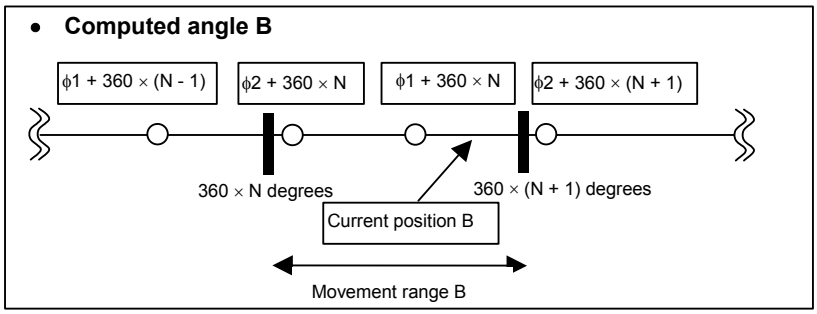


Fig. 22.1 (o) "Computed angle of rotary axis B and its current position and movement range"

When the two axes have a positional relationship as shown in the figure, the output angle of rotary axis A is (θ2 + 360 × N) degrees and that of rotary axis B is (ϕ2 + 360 × N) degrees (when bit 5 (PRI) of parameter No.19608 is set to 0).
More concretely, from the computed angles obtained for rotary axis A, the nearest angle within the movement range, i.e. θ2 + 360 × N degrees, is first adopted. Then, from the computed angles obtained for rotary axis B, the angle belonging to the same group as θ2, i.e. ϕ2 + 360 × N, is adopted.

Note that, in this example, the output angles and moving direction differ depending on whether the movement range is specified or not (0 to 360 degrees), even if N is set to 0 and coordinates are rounded to 0 to 360 degrees.
Namely, if the movement range is not specified, θ1 + 360 degrees nearest to the current position is adopted as the computed angle for rotary axis A and, from the computed angles belonging to the same group as θ1, ϕ1 degrees nearest to the current position is adopted as the computed angle for rotary axis B. Rotary axis A moves in the plus direction. As its coordinate is rounded to 360 degrees, rotary axis A reaches θ1 degrees while moving in the plus direction.
By contrast, when the movement range is set to 0 to 360 degrees, the output angles are (A θ2 degrees; B ϕ2 degrees). Neither rotary axis A nor B moves in a way that it exceeds 0 degree (360 degrees).

- Using tool center point control together with tilted working plane indexing

Tilted working plane indexing (G68.2, G68.3 and G68.4) and tool center point control can be used at the same time.
Please note the following items.

Commanding the motion of rotary axes

Generally, it is impossible to command the motion of rotary axes during tilted working plane indexing.
It is possible in case that tilted working plane indexing and tool center point control are used at the same time. But in that case, it is impossible to command tool axis direction control (G53.1 and G53.6).

Tool direction conversion

Tool direction is converted in case that tilted working plane indexing and tool center point control are used at the same time.

That is, the rotary axes positions are converted so that machining can be performed in the tool direction for the workpiece considered in the program coordinate system (feature coordinate system).

In the start block of tool center point control during tilted working plane indexing, or in the start block of tilted working plane indexing during tool center point control, the rotary axis does not move mechanically and the absolute coordinates are changed by the amount corresponding to the conversion. The axis moves to the specified position in the program coordinate system (feature coordinate system) by the next absolute command. Therefore, an absolute command is necessary after tool center point control starts being used together with tilted working plane indexing.

Depending on the machine configuration, it is physically impossible to turn the tool to the converted direction. In this case, alarm DS0030(TOOL DIRC CMP IMPOSSIBLE) occurs.

In the start block of tool center point control during tilted working plane indexing, or in the start block of tilted working plane indexing during tool center point control, the absolute coordinate values of the rotary axes change considering coordinate system conversion. However, there may be no rotary axis for turning the tool to the tool direction viewed from the program coordinate system (feature coordinate system) depending on the machine configuration. In this case, alarm PS0439 (ILLEGAL COMMAND IN TOOL DIRC CMP) occurs.

Movement of start-up and cancel of Tool length offset

In case that tilted working plane indexing and tool center point control are used at the same time, set bit 6 (TOS) of parameter No. 5006 to 1 (perform tool length compensation by shifting a coordinate system). If parameter TOS = 0, alarm PS0438(ILLEGAL PARAMETER IN TOOL DIRC CMP) is issued.

In case that Tool posture control is used with Type-2 of Tool center point control

In case that tool posture control is used with type-2 of tool center point control, when the angle between a tool posture and the singular posture is less than the parameter No.19738, the tool posture at the end of the block is changed so that the tool posture can pass the singular posture.
But this function is disabled when tilted working plane indexing is used together.

Range of rotary axes movements

In case that type II of tool center point control is used, there are the parameters No.19741 to No.19744 to limit the range of rotary axes movements. When tilted working plane indexing is used together, the rotary axes are converted after the range of rotary axes movements are limited.
Therefore, when the rotary axes are converted, the rotary axes positions may be out of the range of rotary axes movements.
Therefore, in case that the rotary axes have the range of movements, set the stored stroke limit check to prevent a movement over the range.

Functions with which working plane indexing can not be used

Tilted working plane indexing can not be used together with the following functions
- High-speed smooth TCP (G43.4 L1 / G43.4 P3 / G43.5 L1 / G43.5 P3)
- Cutting point command (G43.8 / G43.9)

The order of commands of tilted working plane indexing and tool center point control

There is no restriction about the order of commands of tilted working plane indexing and tool center point control.
But the one commanded later must be canceled earlier. If the one commanded earlier is canceled earlier, alarm PS5421 (ILLEGAL COMMAND IN G43.4/G43.5) occurs.

Manual intervention and handle interruption

During using tool center point control together with tilted working plane indexing, do not perform manual intervention and handle interruption.

Program restart

Do not perform a program restart to the program in which tool center point control is used together with tilted working plane indexing.

Reset

In case that tilted working plane indexing and tool center point control are used at the same time, the tool center point control mode, the tool length compensation vector, and the tilted working plane indexing are canceled by a reset no matter what the setting of the following parameters:
- Bit 6 (CLR) of parameter No.3402
- Bit 0 (C08) of parameter No.3407
- Bit 6 (LVK) of parameter No.5003
- Bit 2 (D3R) of parameter No. 5400.

Workpiece coordinate system shift

Do not use tilted working plane indexing and tool center point control at the same time with G52 (local coordinate system setting) or G92/G50 (T series) (workpiece coordinate system shift) executed.
Once the workpiece coordinate system has been shifted, preset it using G92.1 or axis-specific workpiece coordinate system preset functions before using tool center point control together with tilted working plane indexing.

Parameter INZ

Generally, in case that bit 5 (INZ) of parameter No. 19754 is "0", the workpiece coordinate system is fixed on the table in the state in which tool center point control is started, and subsequently, the coordinate system becomes a table coordinate system.
But during using tool center point control together with tilted working plane indexing, the parameter INZ is regarded as "1" (That is, the workpiece coordinate system is fixed on the table in the state in which the position of the table rotation axis is 0).
And in case that the parameter INZ is "0" and tilted working plane indexing is performed during tool center point control, when tilted working plane indexing is canceled, the workpiece coordinate system is fixed on the table at that time, and subsequently, the coordinate system becomes a table coordinate system.

- Feedrate of rotary axes

In tool center point control, commanded feedrate is regarded as a velocity of tool center point (See EQ 1).

$$dt = \frac{\sqrt{dX^2 + dY^2 + dZ^2}}{F} \qquad \text{(EQ 1)}$$

dt : Time period for motion of a block to take place
F: Commanded feedrate
dX, dY, dZ : Motion distance of the tool center point (X, Y and Z-axis)

In this case, the feedrate of rotary axes is determined as the axes motions of a block ends at the time calculated by EQ1 above.

If rotary axis should be included in commanded feedrate, set parameter ADXx(No.11268#0) for the rotary axis to "1".
If parameter ADXx(No.11268#0) is set to "1", commanded feedrate is regarded as follows.

$$dt = \frac{\sqrt{dX^2 + dY^2 + dZ^2 + dA^2 + ...}}{F} \qquad \text{(EQ 2)}$$

dA, ... : Motion distance of the rotary axis for which the parameter ADXx is set to "1"

Using this dt, the feedrate of tool center point becomes $\frac{\sqrt{dX^2 + dY^2 + dZ^2}}{dt}$.

Operation examples

- In the case of a tool rotation type machine

Explanations are given below assuming a machine configuration in which a tool rotation axis that turns around the Y-axis is located beneath another tool rotation axis that turns around the Z-axis. (See Fig. 22.1 (p).)

If linear interpolation is specified for the X-, Y-, and Z-axes when a workpiece coordinate system is used as the programming coordinate system, control is exerted in such a way that the tool center point moves along a specified straight line with respect to the table (workpiece) as the tool rotates.
Also, speed control is exerted so that the tool center point moves at the specified speed with respect to the table (workpiece).

In the case of a machine having two tool rotation axes, the table does not rotate with respect to the workpiece coordinate system even if the rotary axes move. Therefore, the programming coordinate system always matches the workpiece coordinate system, regardless of whether bit 5 (WKP) of parameter No.19696 is set to 0 or 1.

Examples)
For type 1:

```
O100 (Sample Program1) ;
N1 G00 G90 B0 C0 ;
N2 G55 ;                                  Prepares the programming coordinate system.
N3 G43.4 H01 ;                            Starts tool center point control.
                                          H01 is the tool compensation number.
N4 G00 X200.0 Y150.0 Z20.0 ;              Moves to the start point.
N5 G01 X5.0 Y5.0 Z5.0 C60.0 B45.0 F500 ;  Linear interpolation
N6 G49;                                   Cancels tool center point control.
N7 M30;
```

For type 2:

```
O100 (Sample Program1) ;
N1 G00 G90 B0 C0 ;
N2 G55 ;                                  Prepares the programming coordinate system.
N3 G43.5 H01 ;                            Starts tool center point control.
                                          H01 is the tool compensation number.
N4 G00 X200.0 Y150.0 Z20.0 ;              Moves to the start point.
N5 G01 X5.0 Y5.0 Z5.0 I1.0 J1.732 K2.0 F500 ;  Linear interpolation
N6 G49;                                   Cancels tool center point control.
N7 M30;
```

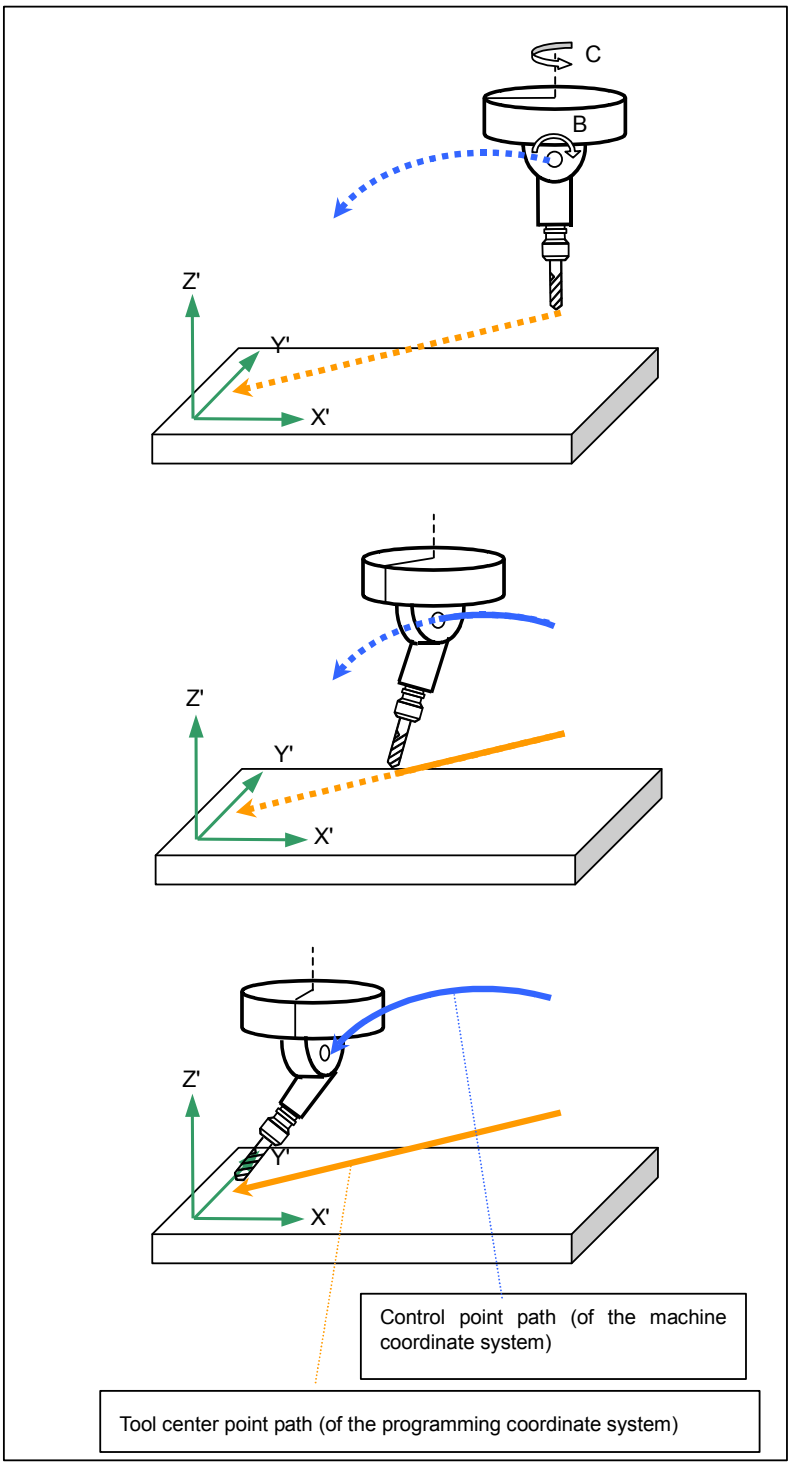


Fig. 22.1 (p) Example for a tool rotation type machine

- In the case of a table rotation type machine

Explanations are given below assuming a machine configuration (trunnion) in which a rotation table that turns around the Y-axis is located above another table rotation axis that turns around the X-axis. (See Fig. 22.1 (q).)

If linear interpolation is specified for the X-, Y-, and Z-axes on the programming coordinate system and if the rotary axis that moves the rotation table is specified (in the case of type 1) or the tool direction is specified (in the case of type 2), control is exerted in such a way that the tool center point moves along a specified straight line with respect to the table (workpiece) as the rotation table rotates.
Also, speed control is exerted so that the tool center point moves at the specified speed with respect to the table (workpiece).

Example)
When type 1 is selected and the coordinate system fixed on the table is used as the programming coordinate system
(Bit 5 (WKP) of parameter No.19696 = 0):

```
O200 (Sample Program2) ;
N1 G00 G90 A0 B0 ;
N2 G55 ;                                        Prepares the programming coordinate system.
N3 G43.4 H01 ;                                  Starts tool center point control.
                                                H01 is the tool compensation number.
N4 G00 X20.0 Y100.0 Z0 ;                        Moves to the start point.
N5 G01 X10.0 Y20.0 Z30.0 A60.0 B-45.0 F500 ;    Linear interpolation
N6 G49;                                         Cancels tool center point control.
N7 M30;
```

When type 1 is selected and the workpiece coordinate system is used as the programming coordinate system
([Bit 5 (WKP) of parameter No.19696 = 1):

```
O200 (Sample Program2) ;
N1 G00 G90 A0 B0 ;
N2 G55 ;                                        Prepares the programming coordinate system.
N3 G43.4 H01 ;                                  Starts tool center point control.
                                                H01 is the tool compensation number.
N4 G00 X20.0 Y100.0 Z0 ;                        Moves to the start point.
N5 G01 X7.574 Y47.247 Z83.052 A60.0 B-45.0 F500 ;  Linear interpolation
N6 G49;                                         Cancels tool center point control.
N7 M30;
```

For type 2 (when the coordinate system fixed on the table is used as the programming coordinate system (only when bit 5 (WKP) of parameter No.19696 is set to 0)):

```
O200 (Sample Program2) ;
N1 G00 G90 A0 B0 ;
N2 G55 ;                                        Prepares the programming coordinate system.
N3 G43.5 H01 ;                                  Starts tool center point control.
                                                H01 is the tool compensation number.
N4 G00 X20.0 Y100.0 Z0 ;                        Moves to the start point.
N5 G01 X10.0 Y20.0 Z30.0 I-1.0 J-2.449 K1.0 F500 ;  Linear interpolation
N6 G49;                                         Cancels tool center point control.
N7 M30;
```

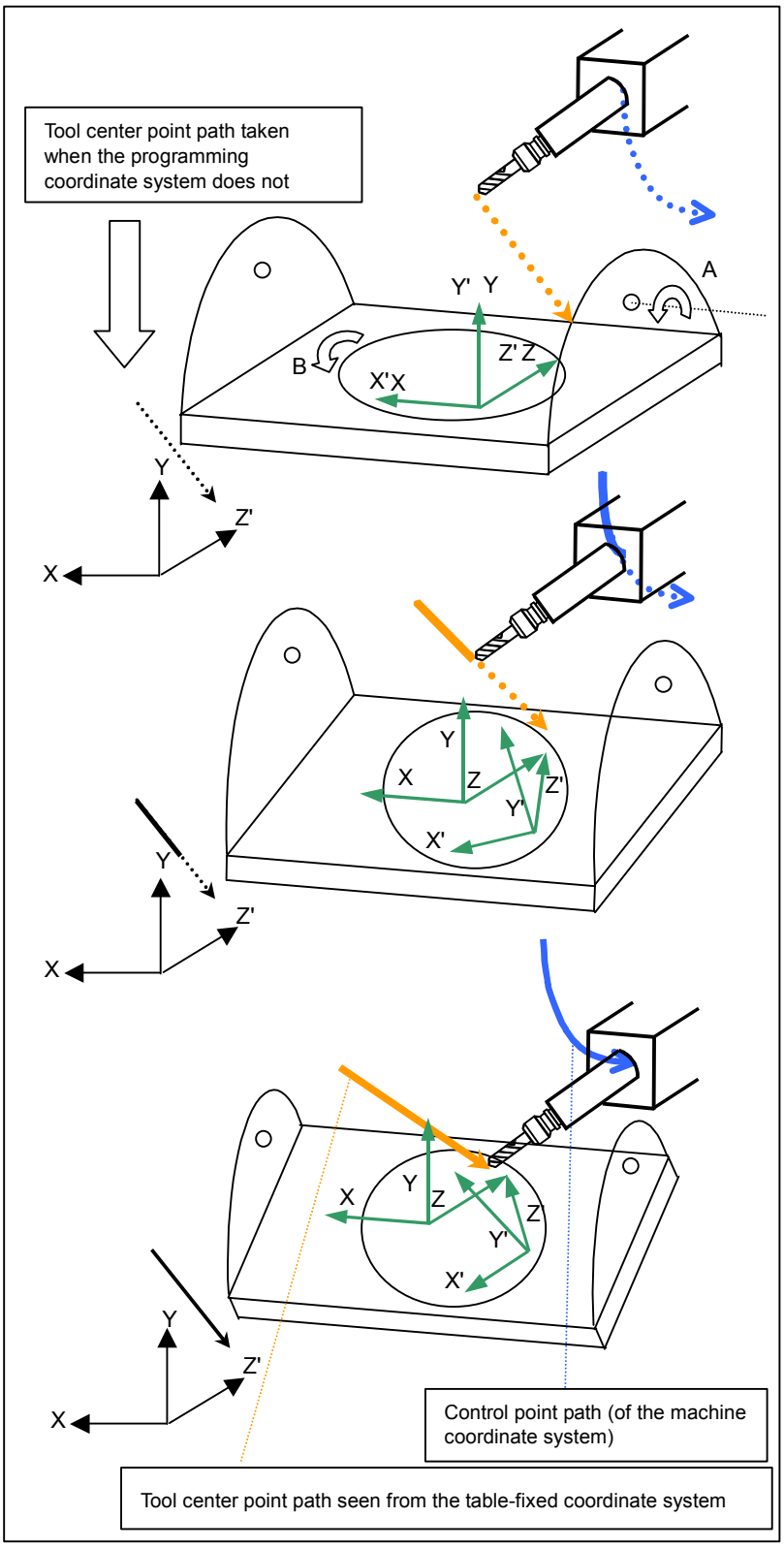


Fig. 22.1 (q) Example for a table rotation type machine

- In the case of a composite type machine

Explanations are given below assuming a composite type machine configuration that has one table rotation axis (which turns around the X-axis) and one tool rotation axis (which turns around the Y-axis). (See Fig. 22.1 (r).)

If linear interpolation is specified for the X-, Y-, and Z-axes on the programming coordinate system and if the rotary axis that moves the rotation table and the tool rotation axis are specified (in the case of type 1) or the tool direction is specified (in the case of type 2), control is exerted in such a way that the tool center point moves along a specified straight line with respect to the table (workpiece) as the rotation table and tool rotate.
Also, speed control is exerted so that the tool center point moves at the specified speed with respect to the table (workpiece).

Example)
When type 1 is selected and the coordinate system fixed on the table is used as the programming coordinate system
(Bit 5 (WKP) of parameter No.19696 = 0):

```
O300 (Sample Program3) ;
N1 G00 G90 A0 B0 ;
N2 G55 ;                                  Prepares the programming coordinate system.
N3 G43.4 H01 ;                            Starts tool center point control.
                                          H01 is the tool compensation number.
N4 G00 X200.0 Y150.0 Z20.0 ;              Moves to the start point.
N5 G01 X5.0 Y5.0 Z5.0 A-60.0 B45.0 F500;  Linear interpolation
N6 G49 ;                                  Cancels tool center point control.
N7 M30;
```

When type 1 is selected and the workpiece coordinate system is used as the programming coordinate system
(Bit 5 (WKP) of parameter No.19696 = 1):

```
O300 (Sample Program3) ;
N1 G00 G90 A0 B0 ;
N2 G55 ;                                  Prepares the programming coordinate system.
N3 G43.4 H01 ;                            Starts tool center point control.
                                          H01 is the tool compensation number.
N4 G00 X200.0 Y150.0 Z20.0 ;              Moves to the start point.
N5 G01 X5.0 Y48.170 Z-79.772 A-60.0 B45.0 F500; Linear interpolation
N6 G49 ;                                  Cancels tool center point control.
N7 M30;
```

For type 2 (when the coordinate system fixed on the table is used as the programming coordinate system (only when bit 5 (WKP) of parameter No.19696 is set to 0)):

```
O300 (Sample Program3) ;
N1 G00 G90 A0 B0 ;
N2 G55 ;                                  Prepares the programming coordinate system.
N3 G43.5 H01 ;                            Starts tool center point control.
                                          H01 is the tool compensation number.
N4 G00 X200.0 Y150.0 Z20.0 ;              Moves to the start point.
N5 G01 X5.0 Y5.0 Z5.0 I2.0 J1.732 K1.0 F500;  Linear interpolation
N6 G49 ;                                  Cancels tool center point control.
N7 M30;
```

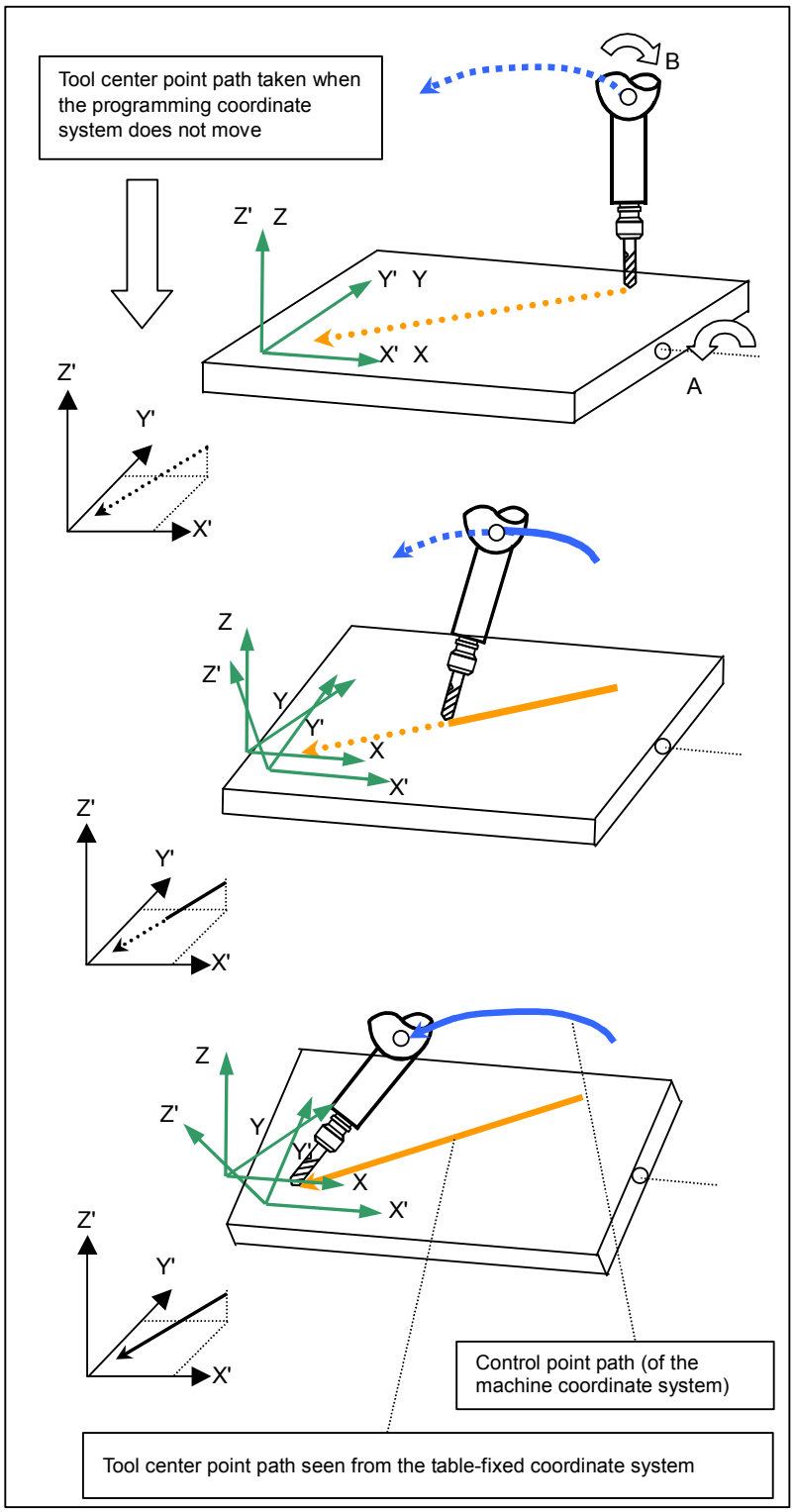


Fig. 22.1 (r) Example for a composite type machine

- When linear interpolation is performed during tool center point control

Examples are given below in which each 100-mm-long side of an equilateral triangle is cut at B-axis angles of 0, 30 to 60, and 60 degrees, respectively.

Example)
When type 1 is selected and the table-fixed coordinate system is used as the programming coordinate system:

```
O400 (Sample Program4) ;
N10 G55 ;                              Prepares the programming coordinate system.
N20 G90 X50.0 Y-70.0 Z300.0 B0 C0 ;    Moves to the initial position.
N30 G01 G43.4 H01 Z20.0 F500. ;        Starts tool center point control.
                                       Moves to the approaching position.
                                       H01 is the tool compensation number.
N40 X28.868 Y-50.0 Z10.0 B30.0 ;       The Z-axis height on the machining plane is 10.0.
N50 Y50.0 ;
N60 B45.0 C120.0 ;
N70 X-57.735 Y0 B60.0 C180.0 ;         Moves X and Y while operating both B- and C-axes.
N80 C240.0 ;
N90 X28.868 Y-50.0 ;
N100 X50.0 Y-70.0 Z20.0 B0 C360.0 ;    X, Y, and Z are approaching positions.
                                       The rotary axes move to their original positions.
N110 G49 Z300.0 ;                      Cancels tool center point control.
                                       Moves the Z-axis to its initial position.
N120 M30;
```

When type 1 is selected and the workpiece coordinate system is used as the programming coordinate system
(Note that the values of N60 to N90 are different from those specified in the preceding example.):

```
O400 (Sample Program4) ;
N10 G55 ;                              Prepares the programming coordinate system.
N20 G90 X50.0 Y-70.0 Z300.0 B0 C0 ;    Moves to the initial position.
N30 G01 G43.4 H01 Z20.0 F500. ;        Starts tool center point control.
                                       Moves to the approaching position.
                                       H01 is the tool compensation number.
N40 X28.868 Y-50.0 Z10.0 B30.0 ;       The Z-axis height on the machining plane is 10.0.
N50 Y50.0 ;
N60 Y-50.0 B45.0 C120.0 ;
N70 X57.735 Y0 B60.0 C180.0 ;          Moves X and Y while operating both B- and C-axes.
N80 X28.868 Y-50.0 C240.0 ;
N90 Y50.0 ;
N100 X50.0 Y-70.0 Z20.0 B0 C360.0 ;    X, Y, and Z are approaching positions.
                                       The rotary axes move to their original positions.
N110 G49 Z300.0 ;                      Cancels tool center point control.
                                       Moves the Z-axis to its initial position.
N120 M30;
```

When type 2 is selected and the table-fixed coordinate system is used as the programming coordinate system:

```
O400 (Sample Program4) ;
N10 G55 ;                                  Prepares the programming coordinate system.
N20 G90 X50.0 Y-70.0 Z300.0 B0 C0 ;        Moves to the initial position.
N30 G01 G43.5 H01 Z20.0 F500. ;            Starts tool center point control.
                                           Moves to the approaching position.
                                           H01 is the tool compensation number.
N40 X28.868 Y-50.0 Z10.0 I1.0 K1.732 ;     The Z-axis height on the machining plane is 10.0.
N50 Y50.0 ;
N60 I-0.5 J0.866 K1.0 ;
N70 X-57.735 Y0 I-1.732 K1.0 ;             Moves X and Y while operating both B- and C-axes.
N80 I-0.866 J1.5 K1.0 ;
N90 X28.868 Y-50.0 ;
N100 X50.0 Y-70.0 Z20.0 K1.0 ;             X, Y, and Z are approaching positions.
                                           The rotary axes move to their original positions.
N110 G49 Z300.0 ;                          Cancels tool center point control.
                                           Moves the Z-axis to its initial position.
N120 M30;
```

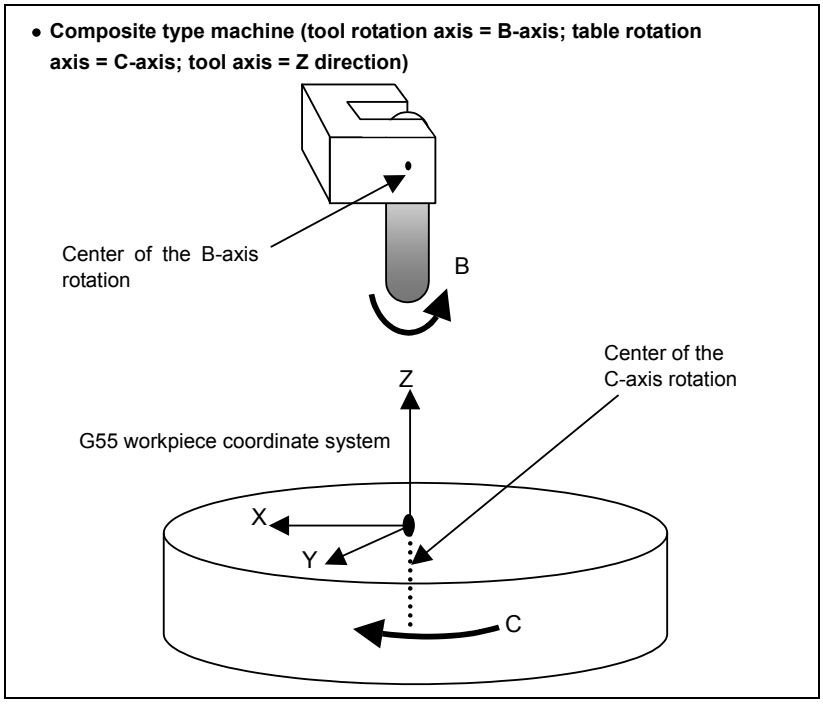


Fig. 22.1 (s) Machine configuration for the example

The following figure illustrates the position of the workpiece, as well as the position of the tool head (relative to the workpiece), as seen from the table-fixed programming coordinate system in the +Z direction.

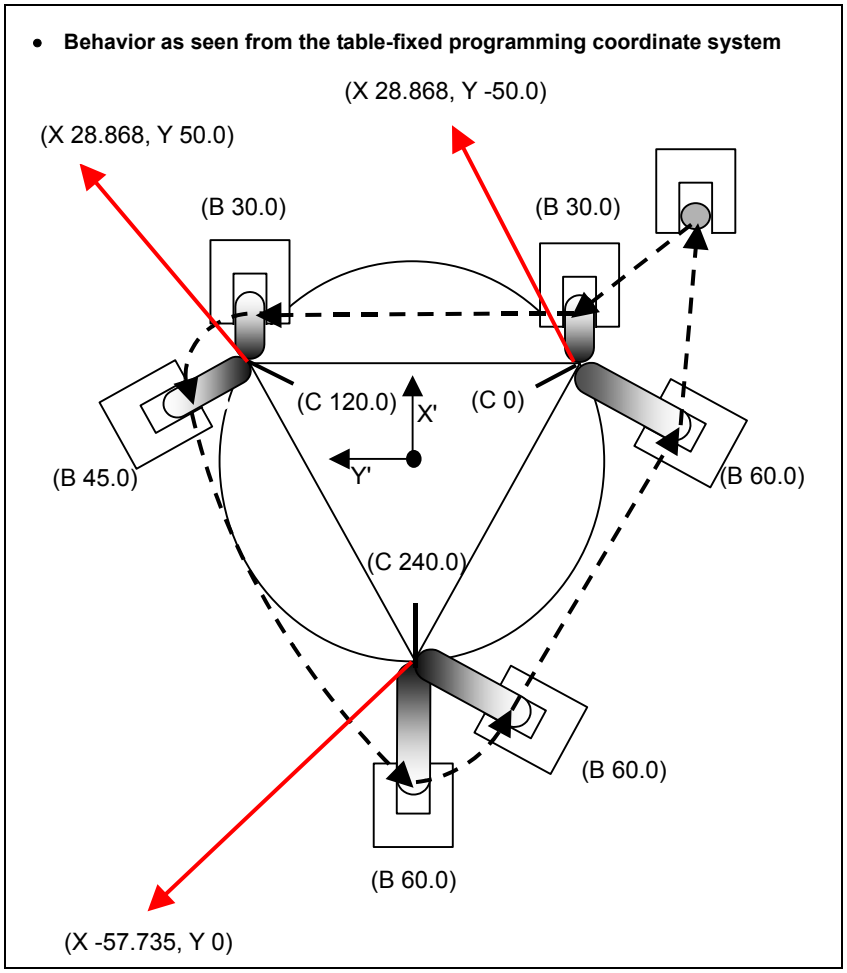


Fig. 22.1 (t) Illustration of the example (1)

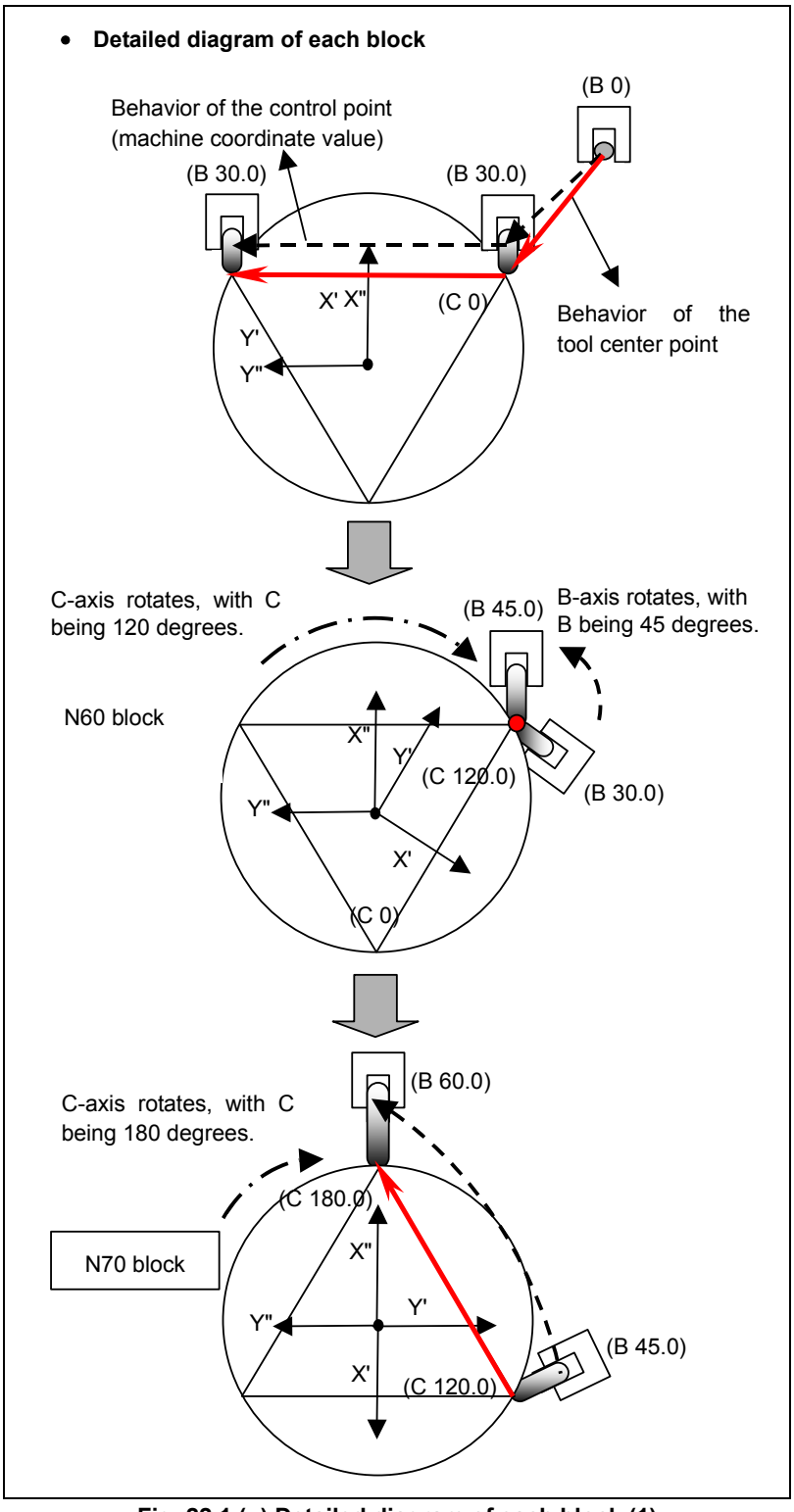


Fig. 22.1 (u) Detailed diagram of each block (1)

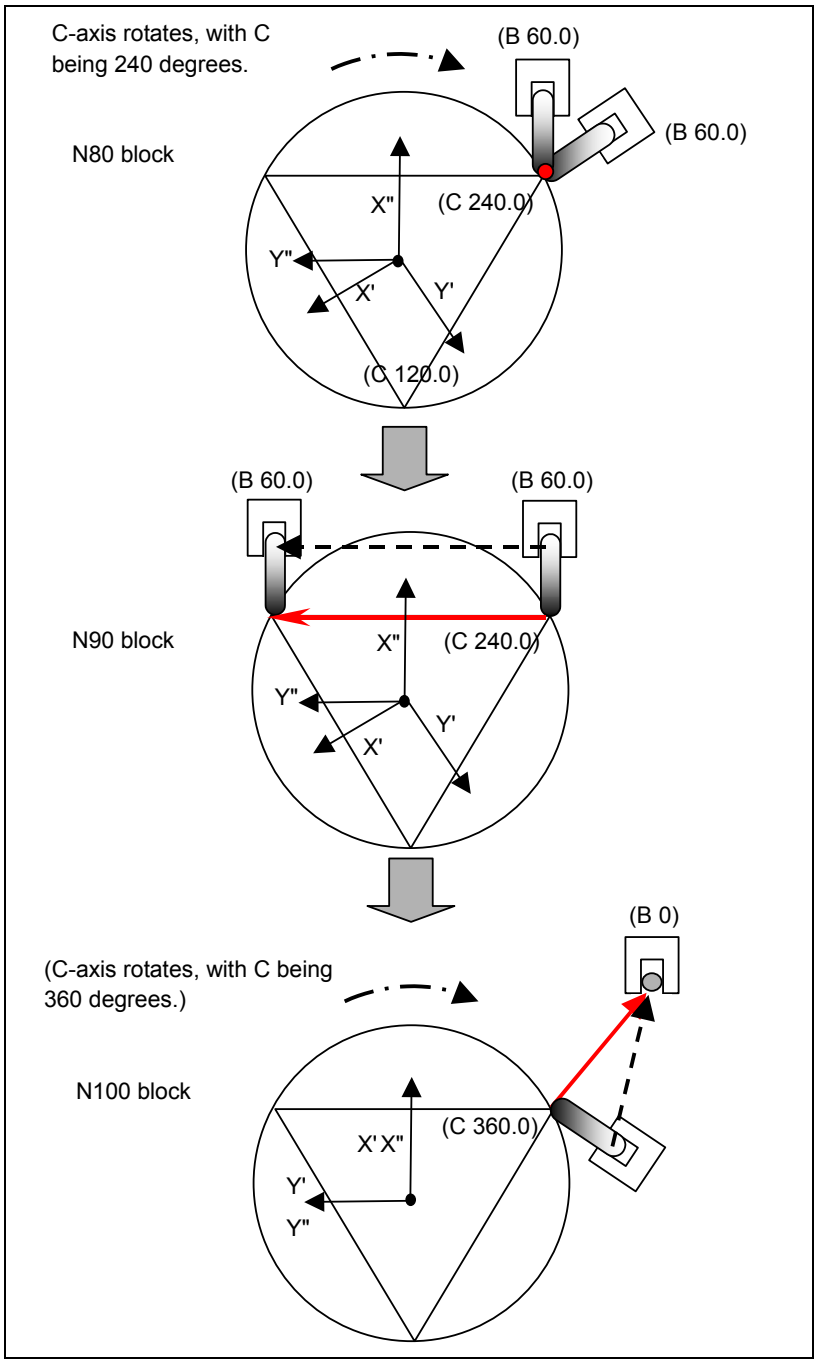


Fig. 22.1 (v) Detailed diagram of each block (2)

- When circular interpolation is performed during tool center point control

In this example, one of the three sides of an equilateral triangle, each being 100 mm long side, is specified as a straight line and the other two are specified as arcs, and each side is cut at B-axis angles of -60, -45 to -30, and -30 degrees, respectively. (For the X-axis, its radius is specified.)
Example)
The program sample given below assumes that the table-fixed coordinate system is used as the programming coordinate system. When the workpiece coordinate system is used as the programming coordinate system (when bit 5 (WKP) of parameter No.19696 is set to 1), the same shape can be obtained by specifying the values shown in parentheses.

```
N001 T0000 ;                                  Cancels the tool offset.
N002 G54 ;                                    Selects the workpiece coordinate system.
N003 G90 X50.0 Y-70.0 Z300.0 B-90.0 C0.0 ;    Moves to the initial position.
                                              The tool tip faces the Z- direction at B-axis angle of -90
                                              degrees.
N020 G01 G43.4 H01                            Starts tool center point control.
N021 Z20.0 ;                                  Moves to the approaching position.
N022 X28.868 Y-50.0 Z10.0 B-60.0 ;            The Z-axis height on the machining plane is 10.0.
N031 Y50.0 ;
N032 B-45.0 C90.0 ;
(N032 X50.0 Y-28.868 B-45.0 C90.0 ;)
N033 G17 G03 X-57.735 Y0.0 J-100.0 B-30.0 C150.0 ;  Moves X and Y while operating both B- and C-axes.
(N033 G17 G03 X50.0 Y28.868 I-100.0 B-30.0 C150.0 ;)
N034 G01 B-30.0 C210.0 ;
(N034 G01 X50.0 Y-28.867 B-30.0 C210.0 ;)
N035 G03 X28.868 Y-50.0 I86.603 J50.0 C270.0 ;
(N035 G03 X50.0 Y28.868 I-100.0 C270.0 ;)
N041 G01 X50.0 Y-70.0 Z20.0 B-90.0 C360.0 ;   X, Y, and Z are approaching positions.
                                              The rotary axes move to their original positions.
N050 G49 ;                                    Cancels tool center point control.
N051 Z300.0 ;                                 Moves the Z-axis to its initial position.
```

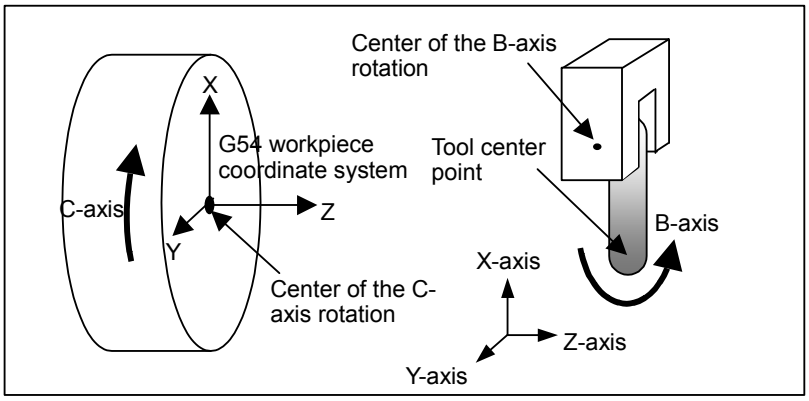


Fig. 22.1 (w) Machine configuration for the circular interpolation example

The following figure illustrates the relative positional relationship between the workpiece and the tool head and their positions, as seen from the +Z direction. The X and Y coordinate values shown in the figure are those on the table-fixed programming coordinate system (which rotates as the C-axis turns).

- **Behavior as seen from the table-fixed programming coordinate system**

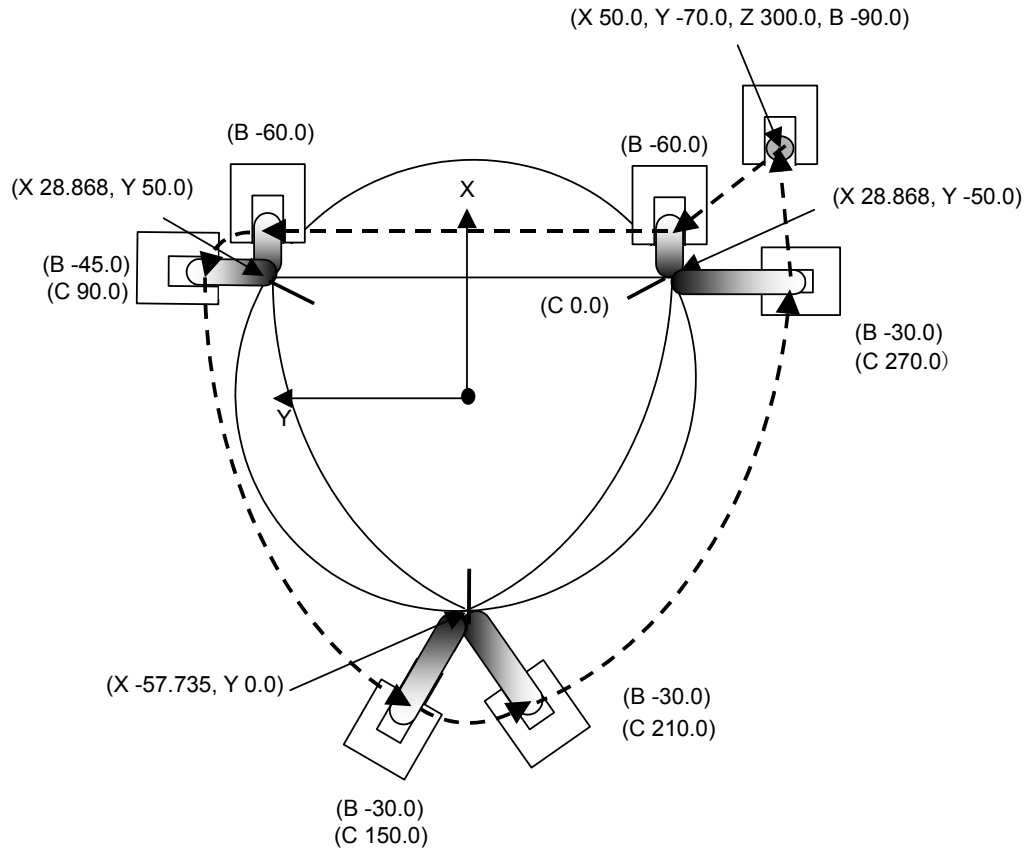


Fig. 22.1 (x) Illustration of the example

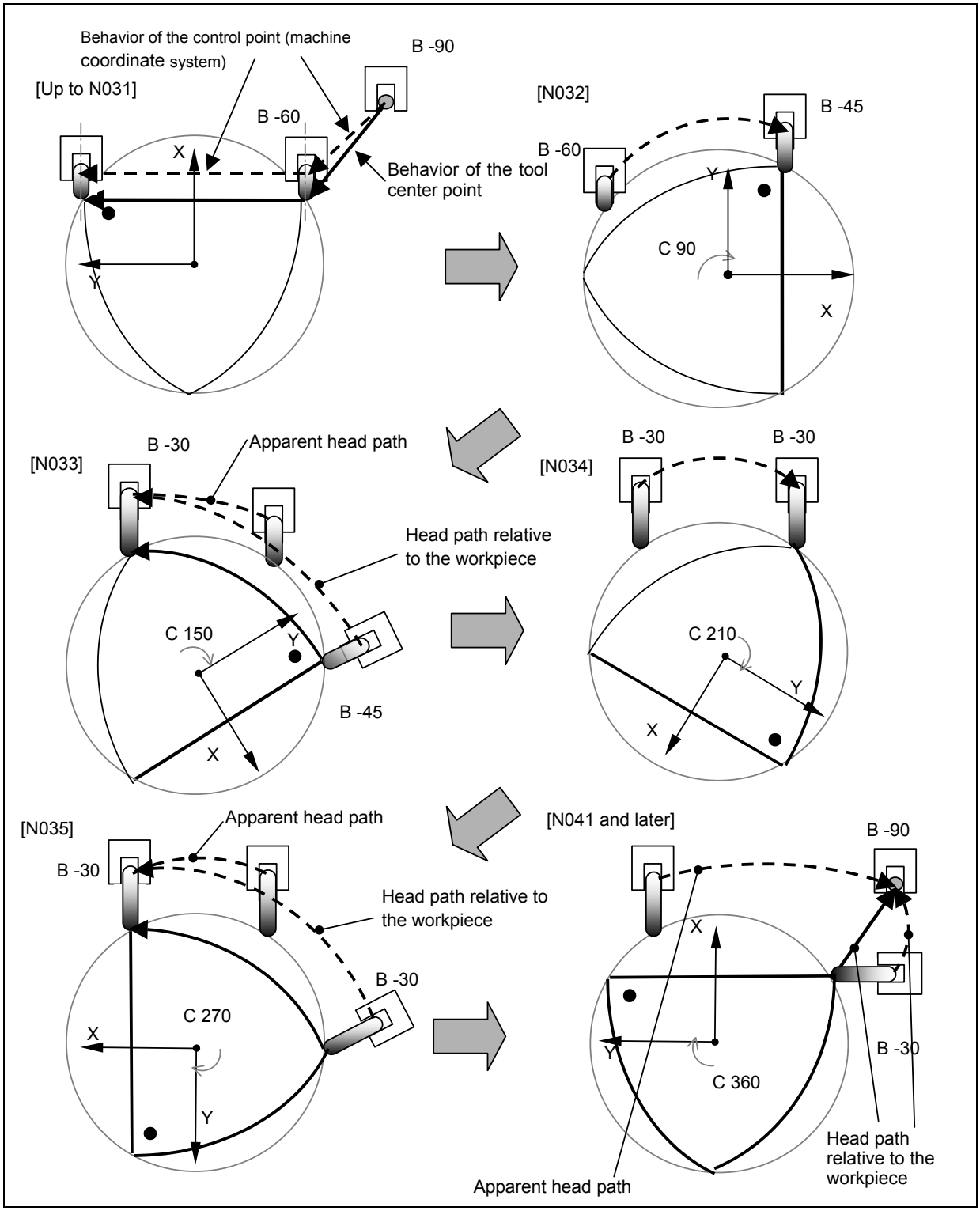


Fig. 22.1 (y) Detailed diagram of each block

Limitation

- Manual intervention to a rotation axis

On a machine of table rotation type or composite type, manual intervention cannot be performed on a table rotation axis during tool center point control. If manual intervention is performed on a table rotation axis on such a machine, alarm PS5421, "ILLEGAL COMMAND IN G43.4/G43.5", is issued at an operation restart.

- Inverse time feed

Manual intervention cannot be performed while inverse time feed is specified during tool center point control. Do not perform manual intervention.

- Setting for workpiece coordinate system (G92) and workpiece coordinate system preset (G92.1)

There are the following limitations when G92 (In case of G code system A in Lathe system, G code is G50.) command or G92.1 (In case of G code system A in Lathe system, G code is G50.3.) command is used during tool center point control.

Limitations common to setting for workpiece coordinate system (G92 or G50) and workpiece coordinate system preset (G92.1 or G50.3):

Mechanical unit type

These G code can be commanded in only tool rotation type (parameter No. 19680 is set to 2).

Specifying to rotation axes

Don't specify to rotation axes. If rotation axes are specified, alarm PS5421, "ILLEGAL COMMAND IN G43.4/G43.5" is issued.

Manual intervention of rotation axes

If manual intervention is used to rotation axes, command a move command to rotation axes before command G92 or G92.1 in absolute mode. If manual intervened rotation axes is specified on G92 or G92.1 block without absolute command, alarm PS5421, "ILLEGAL COMMAND IN G43.4/G43.5" is issued.

Modes that must not be selected

If G92 or G92.1 is commanded in following mode, alarm PS5421, "ILLEGAL COMMAND IN G43.4/G43.5" is issued.

- Tool cutter compensation mode (Other than G40 mode)
- Nano smoothing mode
- Tool posture control mode
- NURBS interpolation mode
- Cutting point command mode

Start-up and cancel of tool center point control

Don't command G92 or G92.1 at a start-up block or cancel block of tool center point control. If G92 or G92.1 is commanded, alarm PS5421, "ILLEGAL COMMAND IN G43.4/G43.5" is issued.

Limitations of setting for workpiece coordinate system (G92 or G50):

Clamp at maximum spindle speed

G92 command (or G50 command) for clamp at maximum spindle speed can't be used. If G92 command (or G50 command) is specified with S command, alarm PS5421, "ILLEGAL COMMAND IN G43.4/G43.5" is issued.

Limitations of workpiece coordinate system preset (G92.1 or G50.3):

Workpiece coordinate system preset functions other than G92.1

Don't use workpiece coordinate system preset functions other than G92.1 command (or G50.3 command). (Example: each axis workpiece coordinate system preset signal)

Bit 7 (WTC) of parameter No. 1205

Even if G92.1 command (or G50.3 command) is specified in tool center point control, tool length compensation vector doesn't clear. (Bit 7 (WTC) of parameter No. 1205 is assumed to be 1.)

- Hypothetical axis of a table rotation axis

When a table rotation axis is set as a hypothetical axis, tool center point control is performed on the assumption that the table rotation axis is at 0 degrees.

- Look-ahead pre-interpolation acceleration/deceleration

When the tool center point control mode is set, look-ahead acceleration/deceleration before interpolation is automatically enabled. Set the parameter for look-ahead acceleration/deceleration before interpolation. If the parameter is not set correctly, alarm PS5420 is issued.

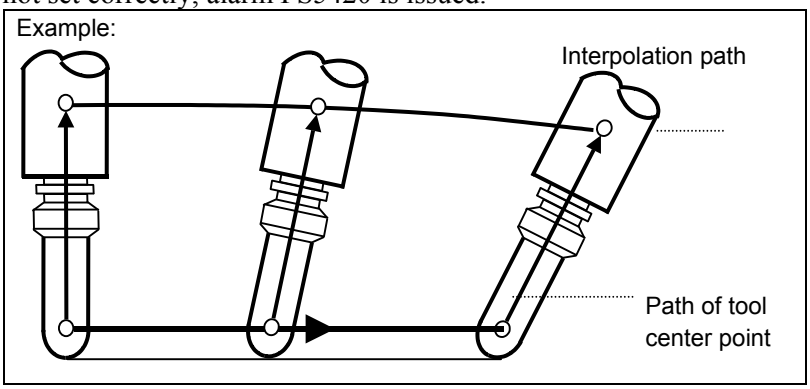


Fig. 22.1 (z)

In the above example, the feedrate is controlled so that the tool center point moves at a specified feedrate, so the feedrate on the interpolation path is equal to or higher than the specified feedrate. At this time, look-ahead acceleration/deceleration before interpolation allows the feedrate to be clamped so that the feedrate on the interpolation path does not exceed the maximum cutting feedrate or maximum rapid traverse rate.

- Deceleration at a corner

Under tool center point control, the controlled point may move along a curve even if a straight-line command is issued. Some commands may cause a corner movement.
When AI contour control is not used, a shock can be applied to the machine at a corner.
So, it is recommended to use AI contour control as well and set an allowable speed difference at a corner (parameter No. 1783) and an allowable acceleration rate (parameter No. 1660, No. 1737).

- 3-dimensional cutter compensation

3-dimensional cutter compensation and tool center point control can be used at the same time.
For restrictions in such a case, see "Restrictions" of 3-dimensional cutter compensation.

- Programmable mirror image

Note the following points when making a programmable mirror image:

- In the case of tool center point control of type 1
  Mirroring the linear axis alone does not create a mirror image for the rotary axis. To make the direction of the tool symmetrical, it is necessary to make a mirror image for the rotary axis as well.
- In the case of tool center point control of type 2
  When bit 6 (MIR) of parameter No.19608 is set to 0, mirroring the linear axis alone does not create a mirror image for I, J, K. Note that the inclination direction specified by Q is the movement direction of the tool center point after mirroring.
  When bit 6 (MIR) of parameter No.19608 is set to 1, mirroring the linear axis automatically creates a mirror image for I, J, K.

A mirror image cannot be put on the rotary axis directly.

- Scaling

Do not apply scaling to a rotary axis.

- Machine coordinate system command

During tool center point control, the machine coordinate system command (G53) cannot be used.
However, G53 can be specified singly to suppress look-ahead operation.

- Stored limit check before move

During tool center point control, stored limit check before move is disabled.

- Type 2

When a type 2 command (G43.5 command) is specified on a machine of table rotation type or composite type (parameter No. 19680 = 12 or 21), the table coordinate system needs to be set as the programming coordinate system (by setting bit 5 (WKP) of parameter No. 19696 to 0). If type 2 is specified with a workpiece coordinate system set as the programming coordinate system, alarm PS5459 is issued.
If a movement area is set for a roll over axis (parameters Nos. 19741 to 19744 ≠ 0), the roll over function is disabled in tool center point control type 2 (G43.5) mode.

- Specifiable G codes

The G codes that can be specified in the tool center point control mode are listed below.
Specifying a G code other than these codes results in alarm PS5421.

- Positioning (G00)
- Linear interpolation (G01)
- Circular interpolation / helical interpolation (G02/G03)
- Dwell (G04)
- AI contour control mode off/on (G05.1 Q0 / Q1)
- Exact stop (G09)
- Programmable data input (G10)
- Tool retract and recover (G10.6)
- Programmable data input mode cancel (G11)
- Plane selection (G17/G18/G19)
- Tool radius compensation cancel (G40)
- 3-dimensional cutter compensation (G41.2/G42.2/G41.4/G42.4/G41.5/G42.5)
- Tool length compensation cancel (G49,G49.1)
- Scaling (G50/G51)
- Programmable mirror image (G50.1/G51.1)
- Machine coordinate system command (G53) specified singly
- Cutting mode(G64)
- Macro call (G65)
- Macro modal call A (G66)
- Macro modal call B (G66.1)
- Macro modal call A/B cancel (G67)
- Tilted working plane indexing (G68.2/G68.3/G68.4)
- Absolute programming (G90)
- Incremental programming (G91)
- Inverse time feed (G93)
- Canned cycle : return to initial level (G98)
- Canned cycle : return to R point level (G99)

M

- Tilted working plane indexing cancel (G69)
- Setting for workpiece coordinate system (G92) (Only tool rotation type)
- Workpiece coordinate system preset (G92.1) (Only tool rotation type)
- Feed per minute (G94)
- Feed per revolution (G95)

T

- Setting for workpiece coordinate system (G50 (G92)) (Only tool rotation type)

- Workpiece coordinate system preset (G50.3 (G92.1)) (Only tool rotation type)
- Tilted working plane indexing cancel (G69.1)
- Feed per minute (G98 (G94))
- Feed per revolution (G99 (G95))

- Modal G codes that allow specification of tool center point control

Tool center point control can be specified in the modal G code states listed below.
Tool center point control cannot be specified in a modal state other than these.

- Modal G codes included in "Specifiable G codes" described previously
- Polar coordinate interpolation mode cancel (G13.1)
- Polar coordinates command cancel (G15)
- Input in inch (G20 (G70))
- Input in mm (G21 (G71))
- Polygon turning cancel (G50.2)
- Workpiece coordinate system setting (G54 to G59)
- Canned cycle cancel (G80)
- Constant surface speed control cancel (G97)

M

- Polar coordinate interpolation mode cancel (G113)

T

- Mirror image for double turret off/balanced cutting mode cancel (G69)

- Specification of axes not relating to tool center point control

If the function of expansion of axis move command in tool center point control is not used, axes not relating to tool center point control cannot be specified. If any is specified, alarm PS5421 is issued.

- Linear axes under tool center point control

The basic three axes set in parameter No. 1022 are regarded as the three linear axes for tool center point control. Axes parallel to the basic three axes cannot be used as the linear axes for tool center point control.
If the basic three axes are not set in parameter No. 1022, alarm PS5459 is issued.

T

- Extended tool selection

When tool center point control is used with a lathe system, the tool change method needs to be ATC type (bit 3 (TCT) of parameter No. 5040 = 1).
In this case, only the value set in "Z" on the tool compensation memory screen is used as a tool length compensation amount. Set values such as "X", "R", and "Y" axis offset values are not used.

22.2 HIGH-SPEED SMOOTH TCP

22.2.1 High-speed Smooth TCP

General

Tool center point control (referred to as TCP in the remainder of this manual) is a 5-axis machining function whereby the tool center point moves along a specified path even if the tool posture varies in relation to the object being machined.
High-speed smooth TCP enables smooth tool center point cutting and tool side cutting while TCP.
High-speed smooth TCP has below two functions.

- Rotation axes compensation (G43.4L1, G43.5L1) :
  G43.4L1/G43.5L1 smoothes the movement of machine compensating the rotation axes to reduce the unevenness in rotation axes commands.
- Smooth control (G43.4P3, G43.5P3) :
  G43.4P3/G43.5P3 interpolates the tool center points and tool postures smoothly.

This function is an option function.

- Rotation axes compensation (G43.4 L1, G43.5L1)

Depending on the machining program, there may be unevenness in tool posture between blocks. In such a case, the position of the control point varies in relation to the position of the tool center point, so that acceleration/deceleration frequently occurs, and the control point does not move smoothly. As a result, there occur increases in machining time, as well as the degradation of the quality of the machined surface (as exemplified by streaks).
High-speed Smooth TCP is a function that compensates rotation axes to reduce the unevenness in tool posture in relation to the tool center path for a machining program for TCP, thereby smoothing the movement of rotation axes.
This is effective to reducing machining time and improving the quality of the machined surface if there is unevenness in tool posture between blocks.
This function is optional.

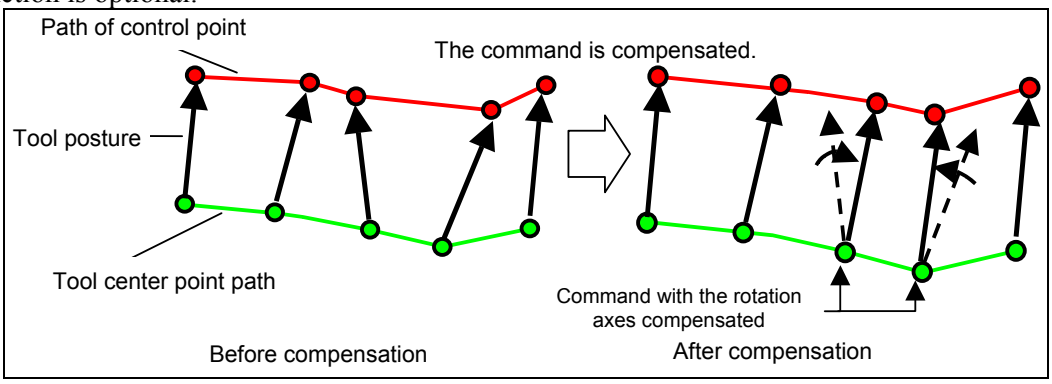


Fig. 22.2.1 (a) Compensation with High-speed Smooth TCP

NOTE
1 The option of "Smooth TCP" or "High-speed Smooth TCP" is necessary to use Rotation axes compensation (G43.4 L1, G43.5L1). It can be used in Series 30*i*/31*i* -B, 31*i* -B5.
2 This function may not be effective to programs and so on that do not involve unevenness in tool posture.
3 In case of tool side cutting which must follow the specified tool posture, this function may not be suitable because the tool posture vector is compensated.

NOTE
4 Each rotation axis is compensated from the original value within the compensation tolerance. The tolerance can be specified by the parameters (No. 10486, 10487) or the G code (G10.8L1).

- Smooth control (G43.4 P3, G43.5P3)

Under tool posture control, the tool posture is controlled every moment for operation on the plane formed by the tool postures (tool length compensation vectors) at the start point and the and point of a block. (Fig.22.2.1 (b))
This control method is suitable for machining a plane with the side of a tool using the tool center point control.

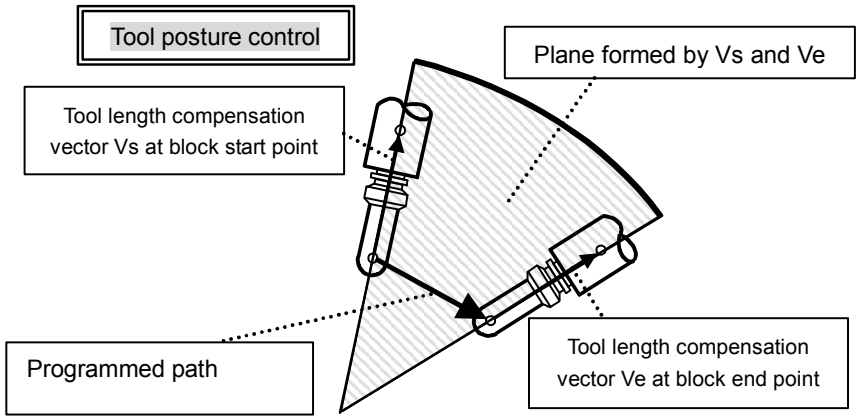


Fig.22.2.1 (b) Tool posture control

Adding to the tool posture control, this feature (High-speed Smooth TCP) generates and interpolates smooth surfaces from commanded tool center points and tool postures.
Traditionally, when a free surface is machined by Tool posture control, the free surface is approximated by polyhedrons. (Fig.22.2.1 (c)) But, by this feature, the free surface can be machined with smooth surfaces and the machining quality can be improved. (Fig.22.2.1 (d))
This feature is suitable for machining that the tool posture is needed to move smoothly using Tool center point control.

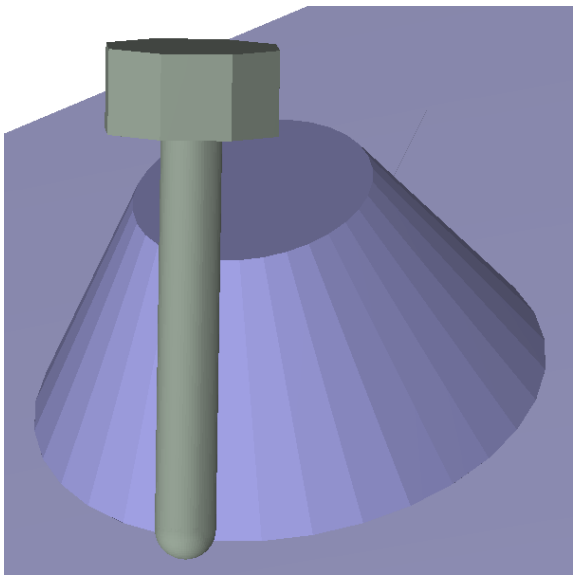


Fig.22.2.1 (c) Polyhedron approximation of free surface

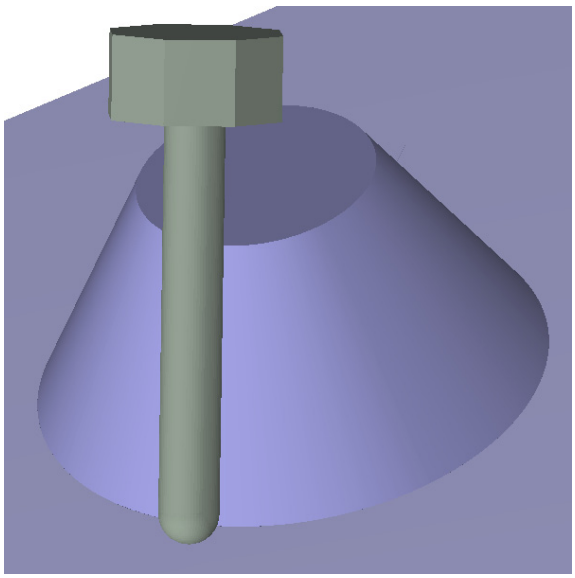


Fig.22.2.1 (d) Free surface machined with this feature

NOTE
The option of "High-speed Smooth TCP" is necessary to use Smooth control (G43.4 P3, G43.5P3). It can be used in Series 30*i* -B/31*i* -B5.

Explanation

There are two types, as described below, one of which is used depending on how the direction of the tool axis is specified.

(1) Type 1
The block end point of the rotary axes is specified (e.g. A, B, C).

(2) Type 2
The direction of the tool axis (I, J, K) at the block end point, as seen from the coordinate system fixed on the table, is specified, instead of the position of the rotary axes.

AI contour control is also turned on with the command G43.4 P3/G43.5P3. Automatic speed control of AI contour control reduces shocks on the machine tool system.

Format

- High-speed Smooth TCP (type 1)

G43.4 L_ P_ IP_ α_ β_ H_ ; (M series) TCP mode is turned ON.
G43.4 L_ P_ IP_ α_ β_ D_ ; (T series) TCP mode is turned ON.
G49 ; TCP mode is turned OFF.

L: 0 : Rotation axes compensation is not effective
1 : Rotation axes compensation is effective
Using bit 0 (STC) of setting parameter No.10485, the state in which the address "L" command is absent can be selected. (Note 1)

P: Selection of Tool posture control mode
0 : Tool posture control is not effective.
1 : Tool posture control is effective.
3 : Smooth control is effective.
(Note 1) Setting parameter TP2(No.11775#0) and TPC(No.19604#0) can select the status in the case that the address P is omitted. (Note 2)

IP: In the case of absolute command, the end position of Tool center point movement
In the case of incremental command, the movement value of Tool center point

α, β: In the case of absolute command, the end position of rotary axes movements
In the case of incremental command, the movement value of rotary axes

H: Tool offset number (M series)

D: Tool offset number (T series)

- High-speed Smooth TCP (type 2)

G43.4 L_ P_ IP_ H_ ; (M series) TCP mode is turned ON.
G43.4 L_ P_ IP_ D_ ; (T series) TCP mode is turned ON.
IP_ I_ J_ K_ ;
G49 ; TCP mode is turned OFF.

L: 0 : Rotation axes compensation is not effective
1 : Rotation axes compensation is effective
Using bit 0 (STC) of setting parameter No.10485, the state in which the address "L" command is absent can be selected. (Note 1)

P: Selection of Tool posture control mode
0 : Tool posture control is not effective.
1 : Tool posture control is effective.
3 : Smooth control is effective.
(Note 1) Setting parameter TP2(No.11775#0) and TPC(No.19604#0) can select the status in the case that the address P is omitted. (Note 2)

IP: In the case of absolute command, the end position of Tool center point movement
In the case of incremental command, the movement value of Tool center point

I,J,K: Tool axis direction at the block end point as seen from the programming coordinate system

H: Tool offset number (M series)

D: Tool offset number (T series)

No rotary axes are specified. Instead, the direction of the tool end point is specified as I, J, K, as seen from the programming coordinate system (the one fixed on the table when G43.5 is specified).
In the case of a tool rotation type machine, I, J, K can be specified in the G43.5 block. But in the case of a table rotation type or composite type machine, I,J,K cannot be specified in the G43.5 block. Specifying I,J,K in the G43.5 block with a table rotation type or composite type machine causes alarm PS2161.
The inclination angle of the tool cannot be specified by using address Q of G43.5. Specifying Q in the G43.5 block causes alarm PS0515.

NOTE

1 In the case that the address L is not commanded, the behavior of G43.4/G43.5 is as follows with the setting parameter STC(No.10485#0).

Parameter STC (No.10485#0)	Behavior of G43.4/G43.5 when address L is not commanded
0	Normal TCP (G43.4L0) starts
1	Rotation axes compensation (G43.4L1) starts

2 In the case that the address P is not commanded, the behavior of G43.4/G43.5 is as follows with the setting parameters TP2(No.11775#0) and TPC(No.19604#0).

Parameter TPC (No.19604#0)	Parameter TP2 (No.11775#0)	Behavior of G43.4/G43.5 when address P is not commanded
0	-	Normal TCP (G43.4P0/G43.5P0) starts
1	0	Tool posture control (G43.4P1/G43.5P1) starts
	1	Smooth control (G43.4P3/G43.5P3) starts

3 To use High-speed Smooth TCP on a T series machine, enable the extended tool selection function (set bit 3 (TCT) of parameter No. 5040 =1). Use D as the tool length offset number. Other restrictions are the same as those on the M series.

4 Alarm PS0515, “ILLEGAL FORMAT IN SMOOTH TCP (G43.4L1)”, occurs if a value other than 0 and 1 is specified for address "L".

5 In the case of simultaneous use of rotation axes compensation with Smooth control, Smoothing is done to the tool postures compensated by Rotation axes compensation.
Therefore, when side cutting is done with this feature, the machined figure is a little different by the angle compensated by Rotation axes compensation from the figure machined without Rotation axes compensation.

Example

A few examples applied to an original program which already exist are shown.
The original program is supposed to be as follows :

```
N040 G43.4 H1 P1 X0.0 Y-100.0 Z0.0;
```
Tool posture control mode is turned ON.

How to enable Smooth control

In order to enable Smooth control, P3 is commanded at the G43.4 block of the original program.

```
N040 G43.4 H1 P3 X0.0 Y-100.0 Z0.0;
```
Smooth control is turned ON.

Or, with the settings of TPC(No.19604#0)=1 and TP2(No.11775#0)=1, P command can be omitted.

```
N040 G43.4 H1 X0.0 Y-100.0 Z0.0;
```
Smooth control is turned ON.

22.2.1.1 Rotation axes compensation (G43.4L1, G43.5L1)

- Compensation of rotation axes

High-speed Smooth TCP is designed to smooth tool posture variations, thereby smoothing the movement of rotation axes.

If there is unevenness in tool posture, the position of the control point varies in relation to the position of the tool center point, so that the control point does not move smoothly and acceleration/deceleration frequently occurs. This results in increases in machining time and the degradation of the quality of the machined surface (as exemplified by streaks).

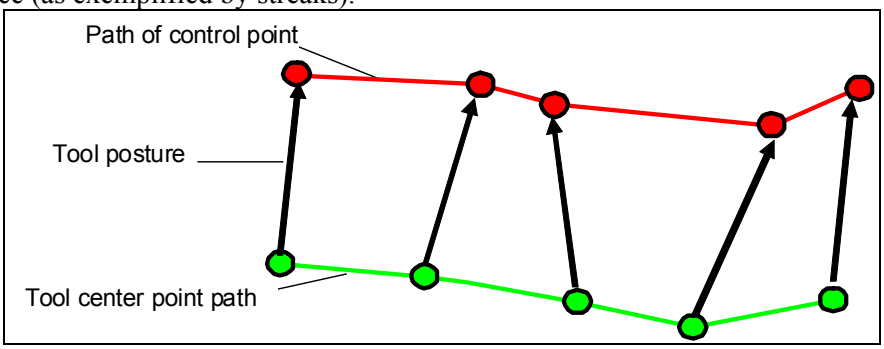


Fig.22.2.1.1 (a) Example of a command that involves unevenness in tool posture

High-speed Smooth TCP does not modify the tool center point position but compensates only the tool posture (rotation axis positions), thereby decreasing the unevenness in tool posture.

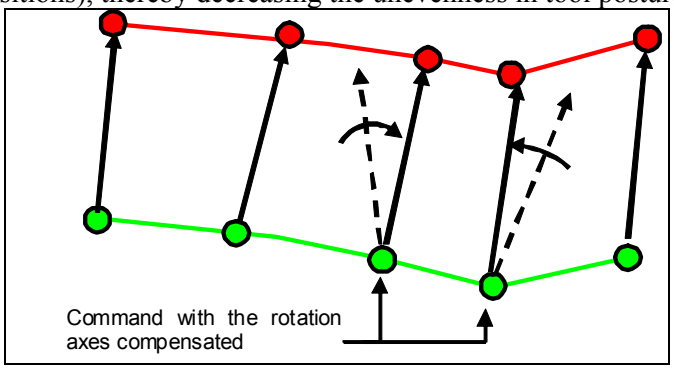


Fig.22.2.1.1 (b) Command after compensation

- Compensation tolerance

High-speed Smooth TCP compensates the rotation axis positions in every block in TCP mode.

Depending on the machining program, as a result of compensation calculations with this function, large compensation may be decided necessary. In such cases, unexpected interferences may be generated between tool and workpiece even for programs that are essentially free from interferences. Moreover, overtravel alarms may occur for linear/rotation axes because of variations in tool posture.

To avoid such large compensation, High-speed Smooth TCP can limit the compensation for the rotation axis positions.

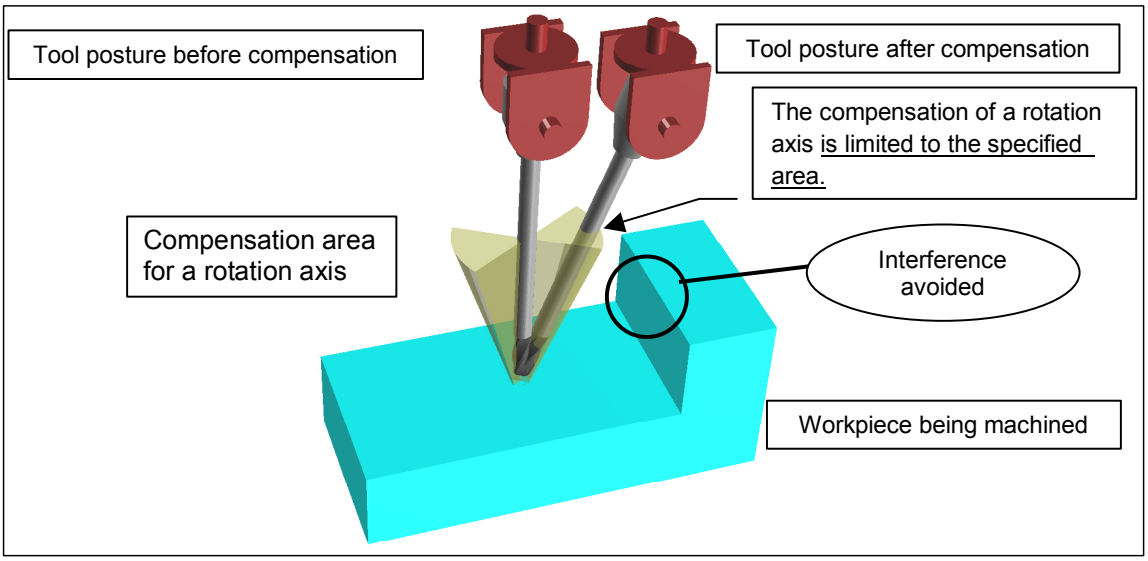


Fig.22.2.1.1 (c) In case of limited compensation

Specify the compensation tolerance for each rotation axis, using parameters Nos. 10486 and 10487.

Setting example)

Parameter No.10486 = 1.0 (First rotation axis compensation tolerance is 1.0 degree.)
Parameter No.10487 = 0.5 (Second rotation axis compensation tolerance is 0.5 degree.)

In the above example, the amount of compensation for the first rotation axis is limited to 1.0 degree and that for the second axis is limited to 0.5 degree.
For example, on a 5-axis machine whose axis configuration is tool head rotation type, where the first rotation axis is a master axis (rotation axis near the workpiece) and the second rotation axis is a slave axis (rotation axis near the tool), the compensation for each rotation axis is limited into the fan-shaped area inFig.22.2.1.1 (d).

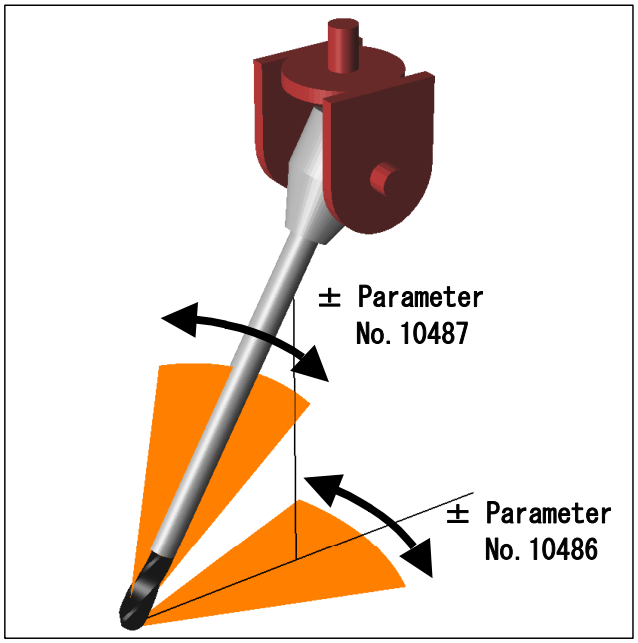


Fig.22.2.1.1 (d) Compensation area for a tool head rotation type machine

NOTE

If the tool posture varies due to compensation with High-speed Smooth TCP while a corner is being machined at the tool center, as in filleting, the finishing allowance may remain uncut at the corner. At such a location, set a small compensation tolerance for each rotation axis so that the tool posture does not vary exceedingly.

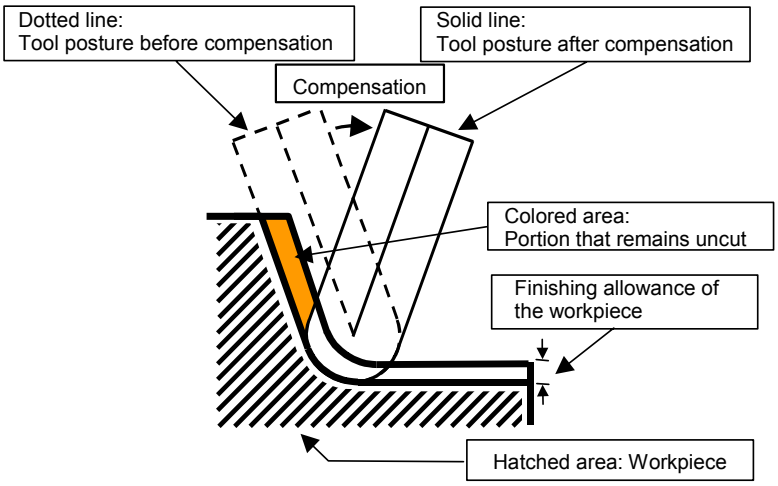


Fig.22.2.1.1 (e)

- Conditions for enabling Rotation axes compensation

High-speed Smooth TCP enables Rotation axes compensation if all of the following conditions are met:

- Block of linear interpolation (G01)
- Block that contains an axis command for any of the five axes
- Block that contains no one shot G code
- Block in which no modal information except for O, N, F, or G code groups 01 and 03 is changed

Rotation axes compensation is canceled in a block that does not meet the above conditions, and it is judged in the next block whether to perform Rotation axes compensation anew.

For example, Rotation axes compensation is canceled if the following blocks are specified.

Example) On a 5-axis machine with three linear axes (X, Y, Z) and two rotation axes (B, C)

```
M55;   (no axis command)
G91 G01;     (no axis command)
G00 X10.0;   (no G01 mode)
G04 X1.0;    (one shot G code)
G51.1 X1.0;  (Address X not being an axis movement command)
```

The position in a High-speed Smooth TCP start block, as well as the end position, is not compensated.
A High-speed Smooth TCP resume block, as well as the single block immediately before High-speed Smooth TCP is canceled, is not compensated, either.
The single block immediately after a High-speed Smooth TCP start block (G43.4L1) is not compensated, either.
The following is an example showing which blocks are compensated after High-speed Smooth TCP starts.
Comments in parentheses to the right of some of the blocks are the reasons why the blocks are not compensated.
The other blocks enclosed in frames are compensated.

Example) On a 5-axis machine with three linear axes (X, Y, Z) and two rotation axes (B, C)

G90 G01 G43.4 L1 H1 Z0 F500 ;	Rotation axes compensation start
G00 X_ Y_ Z_ B_ C_ ;	(Block immediately after Rotation axes compensation starts)
G01 X_ Y_ Z_ ;	(Rotation axes compensation resume block)
Y_ Z_ ;	
X_ Z_ C_ ;	
X_ Y_ ;	(Block immediately before it is canceled)
M55 ;	(No axis command)
X_ Y_ B_ ;	(Rotation axes compensation resume block)
G91 X1.0 ;	
X_ ;	(Block immediately before it is canceled)
G02 X_ Y_ R_ ;	(No G01 mode)
G90 G01 X_ Z_ ;	(Block immediately before it is canceled)
G04 X2.0 ;	(One shot G code)
C_ ;	(Rotation axes compensation resume block)
X_ B_ ;	
B_ C_ ;	
X_ Y_ Z_ C_ ;	(The end position is the same as that before compensation.)
M55 ;	(No axis command)
G49	Rotation axes compensation end

- Insertion of rotation axis commands to blocks in which Rotation axes compensation is enabled

In a block in which Rotation axes compensation is enabled, a rotation axis movement command may be generated due to compensation even if the original command does not have a rotation axis address. The command thus generated is shown in the display of programs under execution.

Example) On a 5-axis machine with three linear axes (X, Y, Z) and two rotation axes (B, C)

Before compensation	After compensation
X_ Y_ ;	X_ Y_ B_ C_ ;
X_ B_ ;	X_ B_ C_ ;
B_ ;	B_ C_ ;

- For machining with a ball end mill

If TCP machining is to be performed with a ball end mill, the tool center point in the program may be specified as the coordinates of the tool tip center or as the coordinates of the tool center point.
At present, High-speed Smooth TCP assumes that for machining with a ball end mill, the program specifies the tool tip center. If the tool tip center is not specified, excessive cutting may occur.

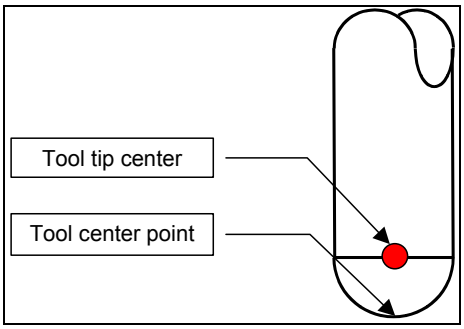


Fig.22.2.1.1 (f) Tip of a ball end mill

Other notes

- Display of programs under execution

In Rotation axes compensation mode, the compensated program is displayed.

- Original program

Rotation axes compensation is not a function to directly edit the original program.
Thus, Rotation axes compensation does not change the contents of the original NC program.

- Machine configurations to which Rotation axes compensation is applicable

This function is available on a 5-axis machine whose axis configuration is head rotation type, table rotation type, or composite type. Also, it is available on either a machining center or lathe machine.

NOTE
Rotation axes compensation is available even on a machine that has only one rotation axis or that uses a hypothetical axis.

- Use with tool posture control

With this function, the tool posture varies within the permissible compensation area. If this function is used together with tool posture control, tool posture control is performed on the tool posture compensated within the permissible area.

Example

The following is an example of applying Rotation axes compensation to an existing machining program for TCP.

Original program

Assume the machining program for TCP (type 1) shown below.
Assume that the machine configuration is such that it contains three linear axes (X, Y, and Z) and two
rotation axes (B as the first and C as the second), with metric input.

```
Example)
     O200
     N010 G49;                          Tool length compensation cancel
     N020 G54 G90 G00 X0 Y0 Z0 B0 C0;   Programming coordinate system specification
     N030 G5.1 Q1;                      AI contour control on
     N040 G43.4 H1 Z0;                  TCP mode on
     N050 G01 X1.0 Y1.0 Z-0.5 B10.0
     C90.0 F1000;
     N060 X2.0 Y0.9 B10.5 C89.5;
     … (Consecutive commands with
     sequences of very small points)
     N880 X1.5 B9.8 C89.0;
     N890 X1.0 B10.0 C90.0;
     N900 G49;                          TCP mode off
     N910 G5.1 Q0;                      AI contour control off
     N920 M30;                          Program end
```

Starting Rotation axes compensation

To start the above program in Rotation axes compensation mode, specify L1 in the frame (TCP start block), as shown below.

```
N040 G43.4 L1 H1 Z0;
```
Rotation axes compensation mode ON

Normal TCP starts if L0 is specified.

```
N040 G43.4 L0 H1 Z0;
```
Normal TCP mode ON

If address "L" is omitted, whether to turn on or off Rotation axes compensation is decided with the setting of bit 0 (STC) of parameter No. 10485.

Parameter STC (No.10485#0)	Behavior of G43.4 when address “L” is not commanded
0	Normal TCP (G43.4L0) starts.
1	Rotation axes compensation (G43.4L1) starts.

If performing Rotation axes compensation

To perform Rotation axes compensation, set the parameters below.

Setting example)

Parameter No.10486 = 1.0 (Compensation tolerance for the first rotation axis is 1.0 degree.)

Parameter No.10487 = 0.5 (Compensation tolerance for the second rotation axis is 0.5 degree.)

With the setting examples above, rotation axis compensation is performed with a compensation tolerance of 1.0 degree for the first rotation axis and with a compensation tolerance of 0.5 degree for the second rotation axis.
If 0 is set in parameter No. 10486, the first rotation axis is not compensated.
If 0 is set in parameter No. 10487, the second rotation axis is not compensated.

22.2.1.2 Smooth control (G43.4P3, G43.5P3)

- Interpolation of Tool center point and Tool posture

This feature generates and interpolates smooth surface which passes commanded Tool center points and Tool postures. Using the following program example, this feature is explained.

Example) The 5-axis machine is supposed as follows :
Head rotation type machine (Linear axes(X,Y,Z) Rotary axes(B,C))
Master rotary axis : C-axis(around Z-axis), Slave rotary axis : B-axis(around Y-axis)

```
G90 G01 F1000 ;
N1 G43.4 P3 X0.0 Y-100.0 Z0.0 B15.0 C90.0 ;
N2 X35.3554 Y-70.7107 C135.0 ;
N3 X50.0 Y0.0 C180.0 ;
N4 X35.3554 Y70.7107 C225.0 ;
N5 X0.0 Y100.0 C270.0 ;
G49 ;
```

Tool center points and Tool postures are commanded as follows Fig.22.2.1.2 (a).

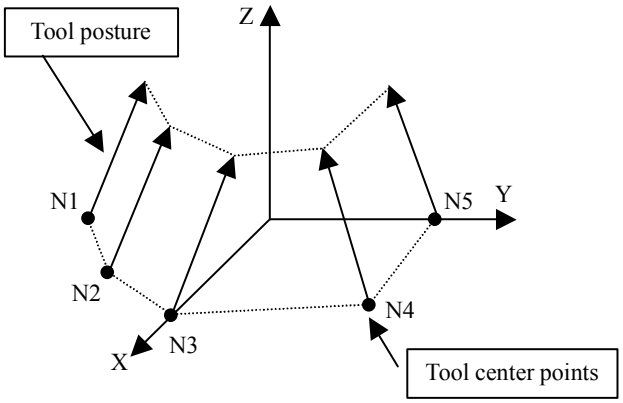


Fig.22.2.1.2 (a) Commanded Tool center points and Tool postures

In case of machining with tool posture control (G43.4P1), the processing surface is shown in Fig.22.2.1.2 (b) (polyhedron).
In case of machining with Smooth control (G43.4P3), the processing surface is shown in Fig.22.2.1.2 (c) (smooth curved surface).

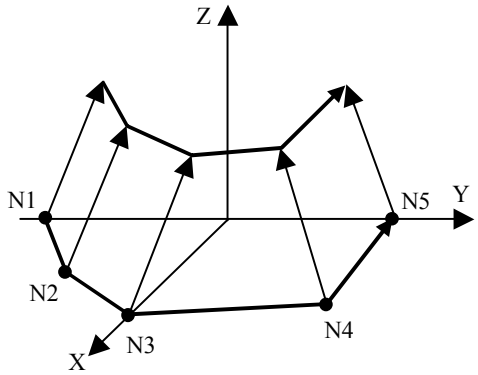


Fig.22.2.1.2 (b) Tool posture control (G43.4P1)

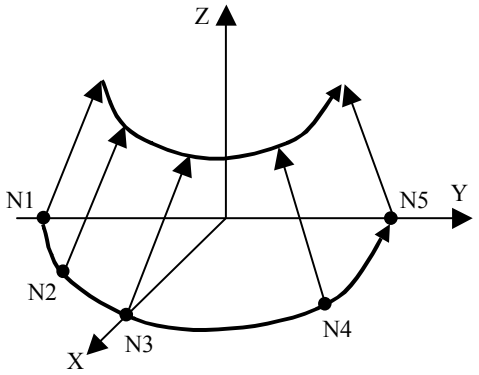


Fig.22.2.1.2 (c) Smooth control (G43.4P3)

- Tolerance of Smooth control

This feature generates and interpolates smooth paths from commanded Tool center points and Tool postures.
Then, the generated surfaces are within the tolerances from the planes generated by the traditional Tool posture control.

(Tool center points)
The generated paths are within the tolerance (Parameter No.11776, or Command G10.8 L2 Q_) from the linear paths generated by the traditional Tool posture control.

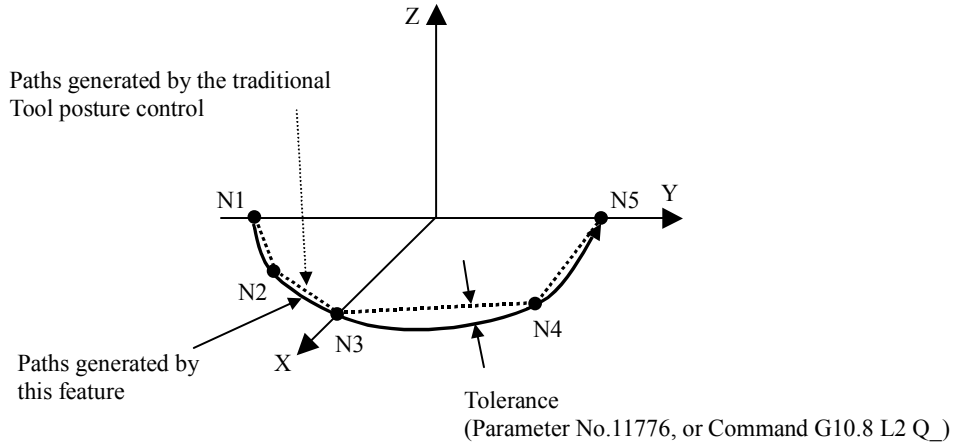


Fig.22.2.1.2 (d) Path of Tool center points

(Tool postures)
The generated tool posture paths are within the angle of tolerance (Parameter No. 11777 or Command G10.8 L2 R_) from the paths generated by tool posture control (G43.4P1).
The closer to 0 the tolerance is, the more the path approaches to the plane generated by tool posture control.
If parameter No.11777 is set to 0, it is assumed that this parameter is set to 0.02.
Usually, it is machined smoothly even if this tolerance is 0.02, so this tolerance does not need to be changed.
When the smoother tool posture path is necessary, set the values (for example, 1.0 degree etc.) to this parameter properly.

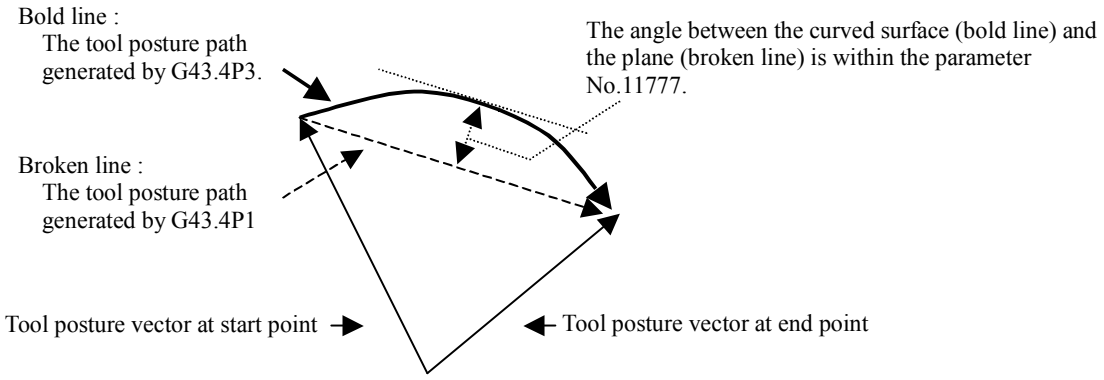


Fig.22.2.1.2 (e) Paths of Tool postures

- Conditions for Smooth control to be effective

Smooth control is effective when the conditions for Tool center point control are satisfied and the following conditions are satisfied.
In the case that the conditions are not satisfied, Smooth control is temporarily canceled and it is decided whether Smooth control should be effective or not at the next block.

Conditions for Smooth control to be effective

1) Modes are as follows.
   - Cutting mode
   - Linear interpolation (G01)
   - Feed per minute (G94, however G98 in G code system A of T series)
   - Polar coordinate interpolation cancel (G13.1)
   - Polar coordinate command cancel (G15)
   - Canned cycle cancel (G80)
   - Scaling cancel (G50)
   - Custom macro modal call cancel (G67)
   - Coordinate system rotation, 3-D coordinate system conversion cancel (G69 (M series), G69.1 (T series))
   - Tool radius compensation cancel (G40)
   - Programmable mirror image cancel (G50.1)
2) One shot G code is not commanded.
3) Buffering is not disabled.
4) Only movements of the axes for Tool center point control are commanded.
5) Block length of Tool center points is less than the parameter No.11778. (When Block length is long enough(longer than the parameter No.11778), it is supposed that Smoothing is not necessary, and Smoothing is temporarily canceled.)

<Path of Tool center point>

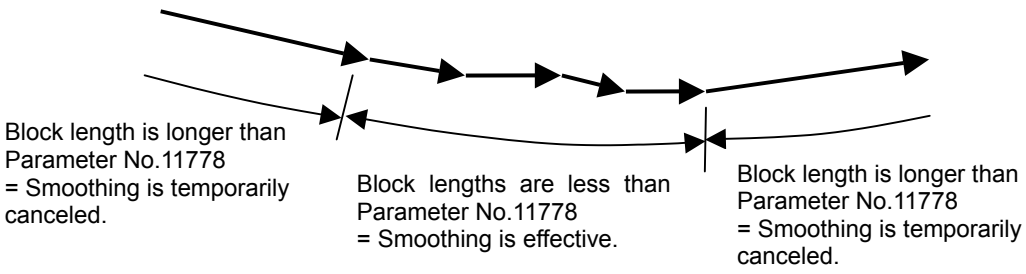


Fig.22.2.1.2 (f) Block length and Smoothing

6) Angle between blocks of Tool center points is less than the parameter No.11779. (In the case angle between blocks of Tool center points is larger than the parameter No.11779, the blocks are supposed to be a corner, and Smoothing is temporarily canceled.)

<Path of Tool center point>

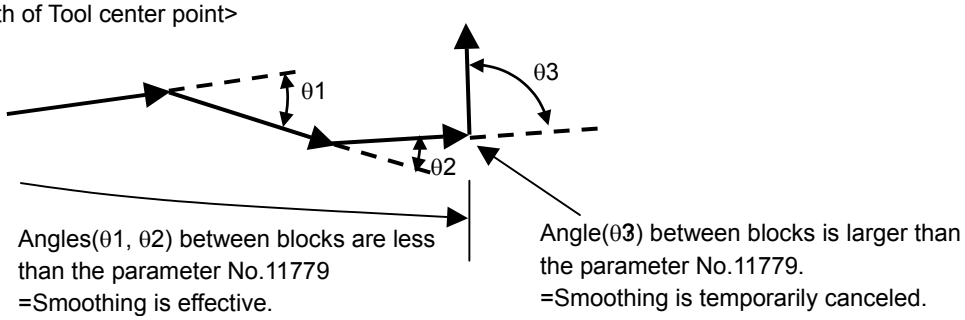


Fig.22.2.1.2 (g) Angle between blocks and Smoothing

7) Movement to cross non over a singular position

Movement to cross over a singular position "Tool side rotary axis" and "Work-piece side rotary axis"

In this paragraph, ‘Tool side rotary axis’ and ‘Work-piece side rotary axis’ which are used in the next paragraph are explained.

On a 5-axis machine, one rotary axis tilts the tool against the work-piece. This rotary axis is called "Tool-side rotary axis", and the other is called "Work-piece side rotary axis".

According to the mechanical unit type, they are as Table 22.2.1.2 (a):

Table 22.2.1.2 (a) Work-piece side rotary axis and Tool side rotary axis

Mechanical unit type (No.19680)	Tool side rotary axis	Work-piece side rotary axis
Tool rotation type (2)	Slave axis	Master axis
Table rotation type (12)	Master axis	Slave axis
Mixed type (21)	Tool rotation axis	Table rotation axis

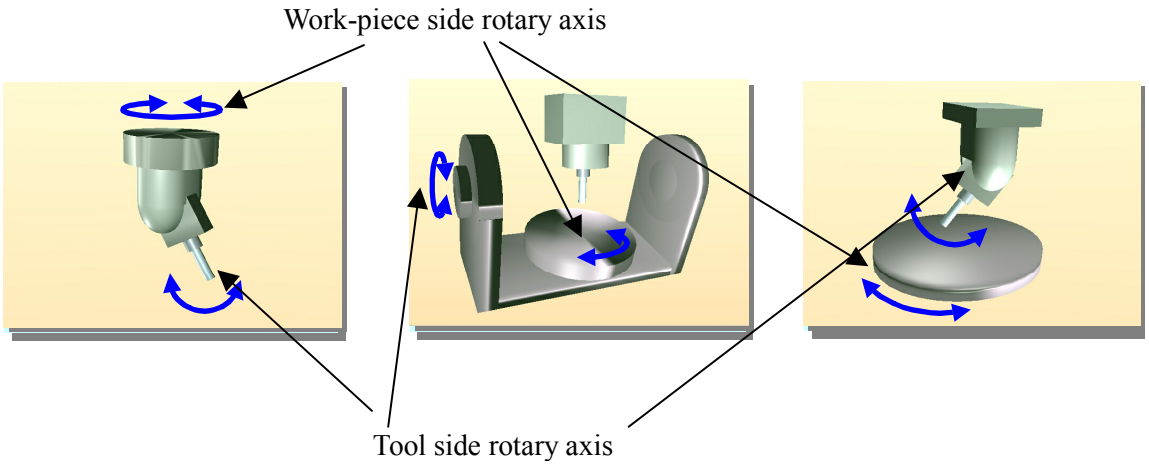


Fig.22.2.1.2 (h) Work-piece side rotary axis and Tool side rotary axis

Singularity, Singular position, Singular posture

A tool posture can be decided by two rotary axes positions. On the contrary, one pair of two rotary axes positions can not be usually decided by a tool posture.
Here, the tool posture where the Work-piece side rotary axis position can be any, in other words, the Work-piece side rotary axis position can not be uniquely decided, is called "Singular posture". And, the position of the Tool side rotary axis at singular posture is called "Singular position (or Singular angle)".
At singular position, the rotary center of Work-piece side rotary axis is parallel to the tool posture (tool direction).

Example)
On a tool rotation type machine which has C axis(master axis around Z axis) and B axis(slave axis around Y axis) and the reference tool direction in Z axis direction, the singular position is B=0,+/-180,,,,deg. Then, the tool posture is the singular posture at C=any-deg.
When B=0 (singular position), the tool posture is a singular posture at C=any-deg., as shown in Fig.22.2.1.2 (i) for example.

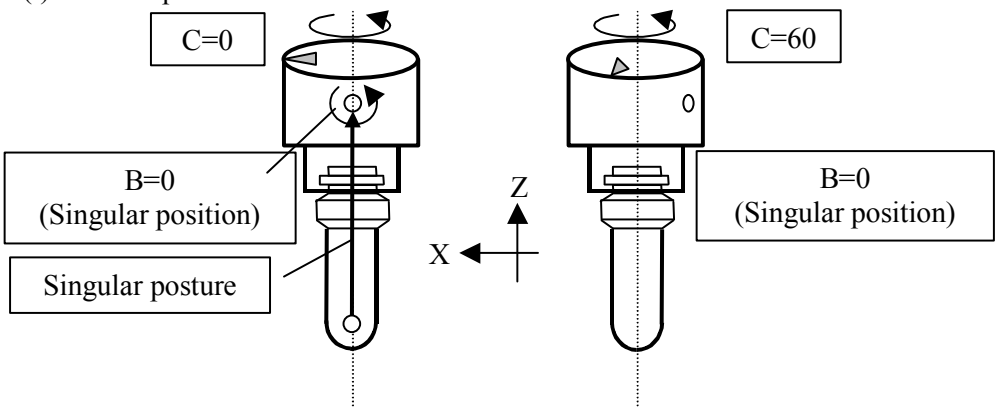


Fig.22.2.1.2 (i) Singular position and Singular posture

Movement to cross over a singular position

When the commanded positions of the tool side rotary axis are opposite in relation to the singular position at the starting point and at the ending point of a block, in other words, when a movement to cross over a singular position is commanded, tool posture control is impossible, and Smooth control (G43.4P3) is also impossible. In this case, Smooth control is canceled temporarily before this command, and it is executed as normal Tool center point control.

Example)
In the following case, Smooth control (G43.4P3) is temporarily canceled (linearization) in 1 block of before and after this command.
- Machine configurations
  : the tool rotation type, master is C-axis (around about Z-axis), slave is B-axis (around about Y-axis)
    the reference tool axis is Z-axis direction, non inclination of reference tool axis direction and rotary axis
- Command
  : rotary angle of start block : (B,C)=(90deg, 0deg) (tool posture is X-axis direction)
  : rotary angle of end block : (B,C)=(-90deg, -90deg) (tool posture is Y-axis direction)

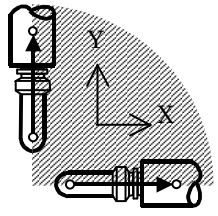
In case, the rotary axis nearest tool is B axis, the singular position is B=0 degree.
Because the start point is B=90 degree and the end point is B=-90 degree, the instruct point of the rotary axis nearest tool (B axis) is placed the singular position and the other side.
The posture (B,C)=(-90deg, -90deg) can not be taken with the tool posture kept in the X-Y plane.
The command (B,C)=(90deg, 90deg) which shows the same posture with (B,C)=(-90deg, -90deg) can be commanded.

```
O0020
…
G43.4 H1 P1;
…
N10 X_ Y_ Z_ B90.0 C0.0;
N20 X_ Y_ Z_ B-90.0 C-90.0;
…
G49;
M30;
```

```
O0021
…
G43.4 H1 P1;
…
N10 X_ Y_ Z_ B90.0 C0.0;
N20 X_ Y_ Z_ B90.0 C90.0;
…
G49;
M30;
```

The commands in N20 in the both programs O0020 and O0021 have same posture (Y-axis direction). But in O0020, Smooth control (G43.4P3) is canceled temporarily before this command, and it is executed as normal Tool center point control, because the tool posture control is impossible from (B,C)=(90deg, 0deg) (the end of N10). Smooth control (G43.4P3) is also impossible.
Otherwise the tool posture control can work in O0021.

B=90.0, C=90.0
or B=-90.0, C=-90.0



B=90.0, C=0.0

Fig.22.2.1.2 (j) Example for impossible Smooth control on type 1

22.2.2 Tolerance change in High-speed Smooth TCP mode

22.2.2.1 Tolerance change in rotation axes compensation (G43.4L1, G43.5L1)

This function is designed to change the compensation tolerance for each rotation axis in Rotation axes compensation mode (G43.4L1, G43.5L1). This makes it possible to change the tolerance according to the situation in part of the machining program.
There are two ways to change the tolerance, directly specifying a value and using the value set in a parameter. No matter which way is used, if the tolerance is changed to 0, compensation on that axis is interrupted temporarily. Compensation can be resumed by specifying a value other than 0 as the tolerance.

Format

G10.8 L1 α_ β_ ; Directly specify the compensation tolerance for each rotation axis.

G10.8 L1 Pn ; From parameters, select the rotation axis compensation tolerance to use.

α: Compensation tolerance for the first rotation axis
β: Compensation tolerance for the second rotation axis
Pn: Specification code for the compensation tolerance for each rotation axis.
n: 0 to 1
0 (0 is used as the compensation tolerance for each rotation axis = Temporary halt of compensation)
1 (Parameters Nos. 10486 and 10487 are used.)

NOTE
1 The option of "Smooth TCP" or "High-speed Smooth TCP" is necessary to use Rotation axes compensation (G43.4 L1, G43.5L1). It can be used in Series 30*i*/ 31*i* -B, 31*i* -B5.
2 G10.8 is a one shot G code.
3 Rotation axes compensation (G43.4L1, G43.5L1) is cancelled temporarily before and after this G code command.
4 If any of the following is specified in a G10.8L1 block, alarm PS0520, "ILLEGAL FORMAT IN G10.8L1", occurs.
- A command other than the above (except O and N commands) is specified.
- P is specified together with α and β.
- A negative value is specified as α or β.
- A value other than 0 and 1 is specified for n.
- Another G code is specified at the same time.
- G10.8 is specified in Rotation axes compensation mode (G43.4L1, G43.5L1), but L is not specified or the value of L is not 1.

5 If G10.8L1 is specified in either of the following conditions, alarm PS0521, "ILLEGAL USAGE OF G10.8L1", occurs.
- Not in Rotation axes compensation mode (G43.4L1, G43.5L1)
- Not linear interpolation (G01)

6 If G10.8 is specified in a G43.4L1/G43.5L1 block, alarm PS0515 occurs.
7 In continuous operation, no deceleration stop occurs in a G10.8L1 block.
8 In single block operation, a single block stop occurs in a G10.8L1 block.
9 If the way to specify a tolerance directly (G10.8L1α_β_), parameters Nos. 10486 and 10487 are not changed.

NOTE
10 Each of the parameters for rotation axis compensation tolerances is clamped with the setting of parameters Nos. 10490 and 10491 as the upper limit.
11 Even if a tolerance is specified directly, it is also clamped with the setting of parameters Nos. 10490 and 10491 as the upper limit.
12 When Rotation axes compensation (G43.4L1, G43.5L1) starts, the value of parameters Nos. 10486 and 10487 is used as the tolerance for each rotation axis.
13 Compensation can be resumed by specifying a value other than 0 as the tolerance.

- Confirming the rotation axis compensation tolerances in use

The rotation axis compensation tolerances used in the block under execution can be confirmed with the following diagnostic data. The tolerances are displayed with the increment systems of the respective rotation axes.

- Compensation tolerance for the first rotation axis: Diagnostic data No. 6501
- Compensation tolerance for the second rotation axis: Diagnostic data No. 6502

Example

The following are examples in which rotation axis compensation with Rotation axes compensation is temporarily halted and resumed. Comments enclosed in parentheses on the right side of certain blocks are reasons why the blocks are not compensated. Other blocks are compensated.

Example 1) Halt of compensation

On a five axis machine (M series) with three linear axes (X, Y, and Z) and two rotation axes (B and C)

```
G90 G01 G43.4 L1 H1     (Rotation axes compensation starts)
Z0 F500 ;
X_ Y_ Z_ B_ C_;         (Block immediately after the start of Rotation axes
                         compensation)
X_ Y_ Z_
X_ Y_ B_
B_ C_;                  (Block immediately before cancellation)
G10.8 L1 P0;            (One-shot G code / Tolerances for the B- and C-axes are set
                         to 0 degrees.)
X_ C_;                  (Rotation axis compensation is temporarily halted.)
X_ Y_ ;                 (Rotation axis compensation is temporarily halted.)
Z_ B_ C_;               (Rotation axis compensation is temporarily halted.)
B_ C_;                  (Rotation axis compensation is temporarily halted.)
X_;                     (Rotation axis compensation is temporarily halted.)
G10.8 L1 B0.5 C1.0;     (One-shot G code / A tolerance of 0.5 degree is set for the
                         B-axis, and a tolerance of 1.0 degree for the C-axis.)
X_ Z_ C_;               (Rotation axes compensation resumption block)
X_ Y_ ;
X_ Z_ B_ C_;
X_ B_ C_;
X_ Y_ C_;               (Rotation axes compensation end position)
G49                     (Rotation axes compensation ends.)
```

Example 2) Halt of the compensation of a specified axis

On a five axis machine (M series) with three linear axes (X, Y, and Z) and two rotation axes (B and C)

```
G90 G01 G43.4 L1 H1 Z0          (Rotation axes compensation starts)
F500 ;
X_ Y_ Z_ B_ C_;                 (Block immediately after the start of Rotation axes
                                compensation)
X_ Y_ Z_
X_ Y_ B_
B_ C_;                          (Block immediately before cancellation)
G10.8 L1 B0.0;                  (One-shot G code / Compensation of the B-axis is halted.)
X_ C_;                          (Rotation axes compensation resumption block)
X_ Y_ ;
Z_ B_ C_;                       Compensation is halted for the B-axis only
B_ C_;
X_;                             (Block immediately before cancellation)
G10.8 L1 C0.0;                  (One-shot G code / Compensation of the C-axis is also
                                halted.)

X_ Z_ C_;                       (Rotation axes compensation resumption block)
X_ Y_ ;
X_ Z_ B_ C_;                    Compensation is halted for both B- and C-axes.
X_ B_ C_;
X_ Y_ C_;                       (Rotation axes compensation end position)
G49                             (Rotation axes compensation ends.)
```

Example 3) Tolerance changes

On a five axis machine (M series) with three linear axes (X, Y, and Z) and two rotation axes (B and C)

```
G90 G01 G43.4 L1 H1 Z0          (Rotation axes compensation starts)
F500 ;
G10.8 L1 P1;                    (One-shot G code / The values of parameters Nos. 10486 and
                                10487 are set as tolerances.)
X_ Y_ Z_ B_ C_;                 (Rotation axes compensation resumption block)
X_ Y_ Z_
X_ Y_ B_
B_ C_;                          (Block immediately before cancellation)
G10.8 L1 B1.0 C0.5;             (One-shot G code / A tolerance of 1.0 degree is set for the
                                B-axis, and a tolerance of 0.5 degree for the C-axis.)
X_ C_;                          (Rotation axes compensation resumption block)
X_ Y_ ;
Z_ B_ C_;
B_ C_;
X_;                             (Block immediately before cancellation)
G10.8 L1 P1;                    (One-shot G code / The values of parameters Nos. 10486 and
                                10487 are set as tolerances.)
X_ Z_ C_;                       (Rotation axes compensation resumption block)
X_ Y_ ;
X_ Z_ B_ C_;
X_ B_ C_;
X_ Y_ C_;                       (Rotation axes compensation end position)
G49                             (Rotation axes compensation ends.)
```

22.2.2.2 Tolerance change in smooth control (G43.4P3, G43.5P3)

In Smooth control mode, the tolerance for paths of Tool center points (parameter No.11776 equal), and The tolerance for angles changing of Tool posture (parameter No.11777 equal) can be changed.
This makes it possible to change the tolerance according to the situation in part of the machining program.

Format

G10.8 L2 Q_ R_ ; Directly specify the tolerance

G10.8 L2 Pn ; From parameters, select the tolerance to use.

Q: Tolerance for paths of Tool center points
R: Tolerance for angles changing of Tool posture
Pn: Specification code for the tolerance in Smooth control mode (G43.4P3).
n: 0 to 1
0 (0 is used as the tolerance)
1 (Parameters Nos. 11776 and 11777 are used.)

NOTE
1 The option of "High-speed Smooth TCP" is necessary to use Smooth control (G43.4 P3, G43.5P3). It can be used in Series 30*i* -B/31*i* -B5.
2 G10.8 is a one shot G code.
3 Smooth control (G43.4P3) is canceled(linearization) temporarily in 1 block of before and after this G code command.
4 The parameter No.11776 and 11777 is not changed, when an each instruction is executed.
5 If any of the following is specified in a G10.8L2 block, alarm PS0523, "ILL. COMMAND IN HIGH-SPEED S-TCP", occurs.
- P is specified together with Q and R.
- A negative value is specified as Q or R.
- A value other than 0 and 1 is specified for n.
- P and neither Q nor R are specified.

22.2.3 Information Display in High-speed Smooth TCP

The following two types of information are displayed in High-speed Smooth TCP.
- Status display in the High-speed Smooth TCP mode
- Maximum compensation of Rotation axes compensation

- Status display in the High-speed Smooth TCP mode

Character string “S-TCP”, which indicates operation in the High-speed Smooth TCP mode, blinks in the status bar (indicated by <1> in Fig.22.2.3 (a)) from the specification of G43.4L1, G43.4P3, G43.5L1 or G43.5P3 to the specification of G49.
This status display is valid until Rotation axes compensation (G43.4L1, G43.5L1) is temporarily interrupted (G10.8L1P0 or G10.8L1α0β0 is specified, or conditions for rotation axes compensation to be satisfied are not met).
Character string “TCP” blinks while compensation is temporarily interrupted.

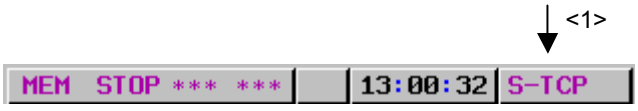


Fig.22.2.3 (a) Status display during execution of High-speed Smooth TCP

Example) Sample S-TCP display

		String displayed in <1>
G90 G01 G43.4 L1 H1 Z0 F500 ;	Smooth TCP mode on	S-TCP
G00 X_ Y_ Z_ B_ C_ ;		S-TCP
G01 X_ Y_ Z_ ;		S-TCP
X_ Z_ C_ ;		S-TCP
G10.8L1B0.0C0.0	Smooth TCP temporarily interrupted (tolerance = 0.0)	TCP
X_ B_ ;		TCP
G10.8L1B1.0C0.8	Smooth TCP restarted	S-TCP
G01 X_ Y_ Z_ C_ ;		S-TCP
X_ Z_ B_ C_ ;		S-TCP
Y_ C_ ;		S-TCP
G49	Smooth TCP mode off	

- Maximum compensation of Rotation axes compensation

In Rotation axes compensation, the larger the specified tolerance, the larger the compensation for a command. If no tolerance is considered, compensation is maximized.
This function indicates the maximum compensation with no tolerance considered for a running program. The function also indicates the maximum compensation applied actually by Rotation axes compensation based on the specified tolerance.
These types of information are displayed based on the calculation results of compensation performed by Rotation axes compensation after the start of Rotation axes compensation (specification of G43.4L1/G43.5L1) until the block that has been looked ahead.
The value of each rotation axis can be checked from the following diagnosis data. The value is indicated in the increment system of each rotation axis.

- Maximum compensation of the first rotation axis with no tolerance considered :
  Diagnostic data No. 6503
- Maximum compensation of the second rotation axis with no tolerance considered :
  Diagnostic data No. 6504
- Maximum value of compensation actually applied to the first rotation axis :
  Diagnostic data No. 6505
- Maximum value of compensation actually applied to the second rotation axis :
  Diagnostic data No. 6506

This data is reset to 0 each time the program of Rotation axes compensation (G43.4L1) starts and continues to be displayed on the diagnosis screen until the next time the program of Rotation axes compensation starts (G43.4L1/G43.5L1 is specified). This data is also reset to 0 when the power is turned on or off.

NOTE
1 The compensation value actually applied is smaller than the maximum compensation with not tolerance considered so as to turn the rotation axis smoothly.
2 The maximum compensation is indicated as an absolute value.

Example)
Display of diagnostic data Nos. 6503 to 6506 (in single block operation)
On a 5-axis machine with three linear axes (X, Y, and Z) and two rotation axes (first rotation axis: A, second rotation axis: C) (machining center system)

(Tolerance setting: For the A-axis (parameter No. 10486): 1.0 degree, for the C-axis (parameter No. 10487): 1.0 degree)

NC statement (before compensation)	NC statement (after compensation)	Compen - sation (A-axis)	Compen- sation (C-axis))	Diagnostic data No.6503	Diagnosis No.6504	Diagnosis No.6505	Diagnosis No.6506	
G90 G00 F500 ;	(No change)							
X0 Y0 Z0 A0 C0 ;	(No change)							
G43.4 L1 H1 Z0 ;	(No change)	0.0	0.0	0.0	0.0	0.0	0.0	(*1)
G00 X1.0 A2.0 C3.0 ;	(No change)							
G01 X3.0 ;	(No change)							
N1 X4.0 C4.0 ;	N1 X4. A2.0623 C3.8369;	+0.0623	-0.1631	0.2	0.8	0.0623	0.1631	(*2)
N2 X6.0 A3.0 ;	N2 X6. A3.1605 C3.9793 ;	+0.1605	-0.0207	0.2	0.8	0.1605	0.1631	(*3)
N3 X7.0 A5.0 ;	N3 X7. A4.6968 C4.0568 ;	-0.3032	+0.0568	1.49	0.8	0.3032	0.1631	(*4)
X9. A5.0 C5.0 ;	(No change)							
G49 ;	(No change)							
G90 G00 ;	(No change)							
X0 Y0 Z0 A0 C0 ;	(No change)							
G43.4 L1 H1 Z0 ;	(No change)	0.0	0.0	0.0	0.0	0.0	0.0	(*1)
G00 X1.0 A1.0 C1.0 ;	(No change)							
G01 X3.0 ;	(No change)							
N4 X4.0 C3.0 ;	N4 X4. A1.0239 C2.6442 ;	+0.0239	-0.3558	0.2	1.6	0.0239	0.3558	(*5)
N5 X6.0 A2.0 ;	N5 X6. A1.9113 C3.0744 ;	-0.0887	+0.0744	0.2	1.6	0.0887	0.3558	(*6)
N6 X7.0 C4.0 ;	N6 X7. A2.1716 C4.0315 ;	+0.1716	+0.0315	0.96	1.6	0.1716	0.3558	(*7)
X9.0 A5.0 C7.0 ;	(No change)							
G49 ;	(No change)							

*1 G43.4L1 is specified and diagnostic data Nos. 6503 to 6506 are reset to 0.

*2 As the result of compensation calculations up to N1 with infinite tolerance considered, a compensation of up to 0.2 degrees is applied to the A-axis and that of up to 0.8 degrees is applied to the C-axis.

*3 As the result of compensation calculations up to N2 with infinite tolerance considered, a compensation of up to 0.2 degrees is applied to the A-axis and that of up to 0.8 degrees is applied to the C-axis.

*4 As the result of compensation calculations up to N3 with infinite tolerance considered, a compensation of up to 1.49 degrees is applied to the A-axis and that of up to 0.8 degrees is applied to the C-axis.
Although the maximum compensation displayed for diagnostic data No. 6503 exceeds the tolerance setting (1.0), the maximum value of compensation actually applied to the A-axis is 0.3032, which is smaller than the tolerance setting (1.0).

*5 As the result of compensation calculations up to N4 with infinite tolerance considered, a compensation of up to 0.2 degrees is applied to the A-axis and that of up to 1.6 degrees is applied to the C-axis.
Although the maximum compensation displayed for diagnostic data No. 6504 exceeds the tolerance setting (1.0), the maximum value of compensation actually applied to the C-axis is 0.3558, which is smaller than the tolerance setting (1.0).

*6 As the result of compensation calculations up to N5 with infinite tolerance considered, a compensation of up to 0.2 degrees is applied to the A-axis and that of up to 1.6 degrees is applied to the C-axis.

*7 As the result of compensation calculations up to N6 with infinite tolerance considered, a compensation of up to 0.96 degrees is applied to the A-axis and that of up to 1.6 degrees is applied to the C-axis.

Diagnose

6501	First rotation axis compensation tolerance in the block under execution

[Data type] Real path
[Unit of data] degree (increment system of the first rotation axis)
[Meaning] The compensation tolerance for the first rotation axis that High-speed Smooth TCP (Rotation axes compensation) is using for compensation in the block under execution is displayed.

6502	Second rotation axis compensation tolerance in the block under execution

[Data type] Real path
[Unit of data] degree (increment system of the second rotation axis)
[Meaning] The compensation tolerance for the second rotation axis that High-speed Smooth TCP (Rotation axes compensation) is using for compensation in the block under execution is displayed.

6503	Maximum compensation of the first rotation axis with no tolerance considered

[Data type] Real path
[Unit of data] degree
[Min. unit of data] Depend on the increment system of the first rotary axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
The maximum compensation of the first rotation axis with not tolerance considered calculated after the start of High-speed Smooth TCP (Rotation axes compensation) until the block that has been looked ahead is displayed as an absolute value.

6504	Maximum compensation of the second rotation axis with no tolerance considered

[Data type] Real path
[Unit of data] degree
[Min. unit of data] Depend on the increment system of the second rotary axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
The maximum compensation of the second rotation axis with not tolerance considered calculated after the start of High-speed Smooth TCP (Rotation axes compensation) until the block that has been looked ahead is displayed as an absolute value.

6505	Maximum compensation actually applied by High-speed Smooth TCP (G43.4L1, G43.5L1) to the first rotation axis

[Data type] Real path
[Unit of data] degree
[Min. unit of data] Depend on the increment system of the first rotary axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
The maximum compensation actually applied to the first rotation axis based on the specified tolerance after the start of G43.4L1 until the block that has been looked ahead is displayed as an absolute value.

6506	Maximum compensation actually applied by High-speed Smooth TCP (G43.4L1, G43.5L1) to the second rotation axis

[Unit of data] degree
[Min. unit of data] Depend on the increment system of the second rotary axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
The maximum compensation actually applied to the second rotation axis based on the specified tolerance after the start of G43.4L1/G43.5L1 until the block that has been looked ahead is displayed as an absolute value.

Other notices

As for other notices, they are subject to Tool center point control.

Restrictions

- Cancellation of High-speed Smooth TCP

High-speed Smooth TCP is canceled with a reset/M02/M30.
For the setting that does not cause TCP to be canceled with a reset, resetting causes High-speed Smooth TCP mode to be canceled, returning the system to normal TCP mode.
The setting that does not cause TCP to be canceled with a reset refers to either of the parameter settings 1 and 2.

Table22.2.3 (a)

Parameter	Bit 6 (CLR) of parameter No. 3402	Bit 0 (C08) of parameter No. 3407	Bit 6 (LVK) of parameter No. 5003
Setting 1	1	1	1
Setting 2	0	—	1

- High-speed Smooth TCP type 2

There are the following limitations.

- The inclination angle of the tool cannot be specified by using address Q. Specifying Q causes alarm PS0515.
- The Cutter Compensation cannot be used together. When not in compensation cancel (G40), specifying the High-speed Smooth TCP (type2) causes alarm PS0515.
- The Arbitrary Angular Axis Control cannot be used together.

Other restrictions are the same as those for tool posture control, High-speed Smooth TCP type1 and tool center point control type2.

NOTE
Even if the tolerance is 0, the path in High-speed Smooth TCP type2 may become different from that in Tool Center Point Control type2.
The difference is about input increment.

- Intervention with an MDI command during a single block stop

Do not intervene with an MDI command during a single block stop in High-speed Smooth TCP mode.
Intervention with an MDI command issues alarm PS5421.

- Manual intervention

Manual intervention cannot be performed during Smooth control (G43.4P3).
If manual intervention is performed, alarm PS0523, "ILL. COMMAND IN HIGH-SPEED S-TCP", is issued at an operation restart.

- Other restrictions

Other restrictions are the same as those for tool center point control.

22.3 EXPANSION OF AXIS MOVE COMMAND IN TOOL CENTER POINT CONTROL

Overview

This function makes it possible to specify commands for axes other than the five axes subject to tool center point control during tool center point control.

Details

In the context of tool center point control, the axes subject to tool center point control are called "5-axis machining control axes", while other axes are called "non 5-axis machining control axes".
If tool center point control is performed with four axes, with a single rotation axis as a hypothetical axis, the four axes are assumed to be 5-axis machining control axes.
This function makes it possible to specify move commands for non 5-axis machining control axes during tool center point control.

The maximum number of non 5-axis machining control axes that can be specified in a single block is 10 in Series 30*i*-B, and 2 in Series 31*i*-B5. If the number of non 5-axis machining control axes exceeds these limits, the alarm PS5421, "ILLEGAL COMMAND IN G43.4/G43.5", occurs.
Tool center point control can be used in Series 31*i*-B, but this function cannot be used in Series 31*i*-B.

- Feedrate of non 5-axis machining control axes

In tool center point control, the feedrate of each axis is determined so that the tool center point moves at the specified feedrate (in the case of G01) or the rapid traverse rate (in the case of G00). In the case of a block in which the rotation axis moves while the tool center point does not move in relation to the workpiece, the feedrate of each axis is determined so that the rotation axis moves at the specified feedrate or the maximum cutting feedrate (in the case of G01) or the rapid traverse rate (in the case of G00).
The feedrate of non 5-axis machining control axes is determined so that the movement of the non 5-axis machining axes ends at the same time as the end of the movement of the block due to the movement of the 5-axis machining control axes at the feedrate described above.
If the feedrate of non 5-axis machining control axes exceeds the maximum cutting feedrate (in the case of G01) or the rapid traverse rate (in the case of G00), the feedrate of all axes is clamped so that it does not exceed.
Note that if only non 5-axis machining control axes are specified in tool center point control mode, the axes move at the same feedrate as that when they are specified when tool center point control is not in progress.

- In case that non 5-axis machining control axes should be included in commanded feedrate

If non 5-axis machining control axes should be included in commanded feedrate, set parameter ADXx(No.11268#0) for the axes to "1".
If parameter ADXx(No.11268#0) is set to "1", commanded feedrate is regarded as follows.

$$dt = \frac{\sqrt{dX^2 + dY^2 + dZ^2 + dU^2 + dV^2 + ...}}{F}$$

dt : Time period for motion of a block to take place
F: Commanded feedrate
dX, dY, dZ : Motion distance of the tool center point (X, Y and Z-axis)
dU, dV, ... : Motion distance of the non 5-axis machining control axis for which the parameter ADXx is set to "1"

Using this dt, the feedrate of tool center point becomes $\frac{\sqrt{dX^2 + dY^2 + dZ^2}}{dt}$.

In case parameter ADF(No.11269#2)=1, all the non 5-axis machining control axes are included in commanded feedrate regardless of the setting of parameter ADXx.

- Interpolation type of non 5-axis machining control axes

The interpolation type of non 5-axis machining control axes is linear interpolation.

Restrictions

- Functions that cannot be used simultaneously

If the following functions are used in tool center point control, no move commands for non 5-axis machining control axes can be specified. If they are specified, alarm PS5421, "ILLEGAL COMMAND IN G43.4/G43.5", occurs.

- Nano smoothing
- NURBS interpolation

- Notes on the number of simultaneously controlled axes

The number of simultaneously controlled axes is limited due to the maximum allowable number of simultaneously controlled axes of the system.

NOTE

1 Circular interpolation cannot be performed on non 5-axis machining control axes.
2 If, in circular interpolation, the amount of movement on a 5-axis machining control axis is 0, the operation is as described below.
   - If an arc radius command (R, I, J, or K) is specified:
     Even if a non 5-axis machining control axis is specified, the axis is ignored. No alarm is issued.
   - If no arc radius command (R, I, J, or K) is specified:
     Alarm output follows bit 5 (CIR) of parameter No. 3403. If, however, only a non 5-axis machining control axis is specified, no alarm is issued even if bit 5 (CIR) of parameter No. 3403 is 1. The axis is ignored.

22.4 TOOL POSTURE CONTROL

Overview

Under tool center point control, the tool tip moves along a specified path even when the tool direction relative to the workpiece changes. Usually, however, the two rotary axes are controlled independently of each other, and the posture of the tool is not controlled.

Under tool posture control, the present tool posture is controlled so that it can operate on the plane made up of the tool postures at the block start and end points (tool length compensation vectors) in positioning or linear interpolation.

This control method is suitable for machining a plane with the side of a tool by using tool center point control.

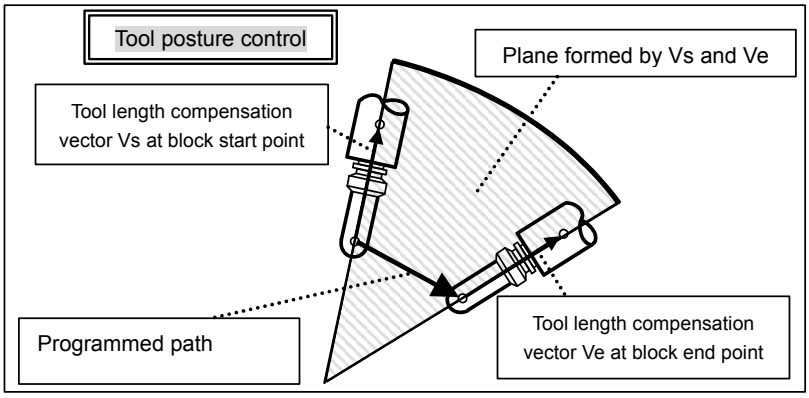


Fig. 22.4 (a) Tool posture control (positioning or linear interpolation)

In circular interpolation and helical interpolation, the tool posture is varied smoothly from the block start point to the end point. By appropriately performing a tool posture command at the block end point, conical side face machining can be performed with a tool side face.

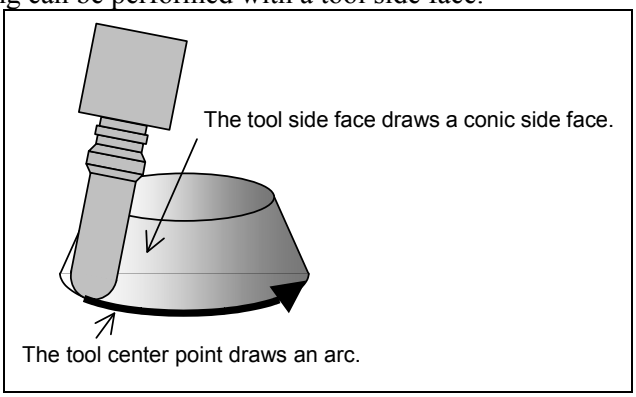


Fig. 22.4 (b) Tool posture control (circular interpolation)

This function is optional. The tool center point control option is also needed.

Format

Format
G43.4 IP_ α_ β_ H_ P_ ; Tool center point control (type 1)
G43.5 IP_ H_ Q_ P_ ; Tool center point control (type 2)
Description of symbols
IP : For an absolute command, the coordinates of travel end point of the tool center point For an incremental command, the travel distance of the tool center point
α, β : For an absolute command, the coordinates of the end point of the rotation axis For an incremental command, the travel distance of the rotation axis
H : Tool offset number
Q : Tool tilted angle (degree)
P : Tool posture control disabled (P0) or tool posture control enabled (P1) However, bit 0 (TPC) of setting parameter No. 19604 can be used to select the behavior taken when the address P command is not present.

Tool posture control is enabled when P1 is specified or disabled when P0 is specified in the G43.4 and G43.5 blocks. The behavior taken when the address P command is not present can be selected by bit 0 (TPC) of parameter No. 19604. The setting of tool posture control selected in the G43.4 and G43.5 blocks is kept until the tool center point control cancellation command (G49) is specified.

Example:

```
O0010
…
N10 G90 G00 X30.0 Y-30.0 Z0.0;
N20 G43.5 H1 P1;
N30 G42.6 D2;
N40 G01 X20.0 Y-20.0 I-1 J1 K2 F_;
N50 Y20.0 I-1 J-1 K2;
N60 X-20.0 I1 J-1 K2;
N70 Y-20.0 I1 J1 K2;
N80 X20.0 I-1 J1 K2;
…
G40;
G49;
M30;
```

In N20 to N80 in program O0010 to the left, the sides of a quadrangular pyramid trapezoid as shown below can be machined by the side of the tool using tool posture control. The following figure illustrates tool movement by N50. The command in N30 is the three-dimensional cutter compensation command.

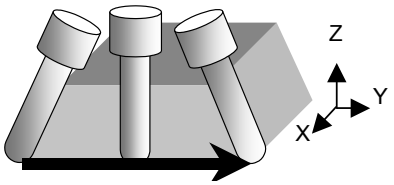


Fig. 22.4 (c) Example of tool posture control (linear interpolation)

Example:

```
O0010
…
N10 G90 G00 X0 Y0 Z0;
N20 G43.5 H1 P1;
N30 G01 I1 J0 K2 F_;
N40 G17 G02 X20.0 Y20.0 R20.0 I0
J-1 K2;
…
G40;
G49;
M30;
```

In N40 in program O0010 on the left, (part of) the side of a conical shape such as that shown below can be machined with the side of the tool, using tool posture control.

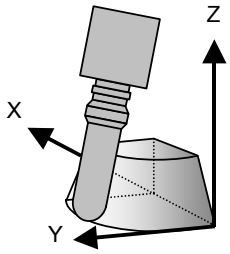


Fig. 22.4 (d) Example of tool posture control (circular interpolation)

Explanation

In tool posture control (positioning or linear interpolation), the two rotation axes are controlled so that the momentary tool posture (tool length compensation vector) satisfies the following conditions (Fig. 22.4 (e)).
The momentary tool posture (tool length compensation vector) is present on the plane configured by the tool length compensation vector at the block start point and that at the block end point.
The angle which the tool length compensation vector at the block start point makes with the momentary tool length compensation vector is the angle obtained by proportionally dividing the angle which the tool length compensation vector at the block start point makes with the tool length compensation vector at the block end point.

Tool posture control is enabled when P1 is specified or disabled when P0 is specified in the tool center point control start command (G43.4/G43.5) blocks. Bit 0 (TPC) of setting parameter No. 19604 can be used to select whether tool posture control is enabled or disabled when the address P command is not present. The setting of tool posture control selected in the G43.4 and G43.5 blocks is kept until the tool center point control cancellation command (G49) is specified.

Other notes and restrictions are the same as those for tool center point control.

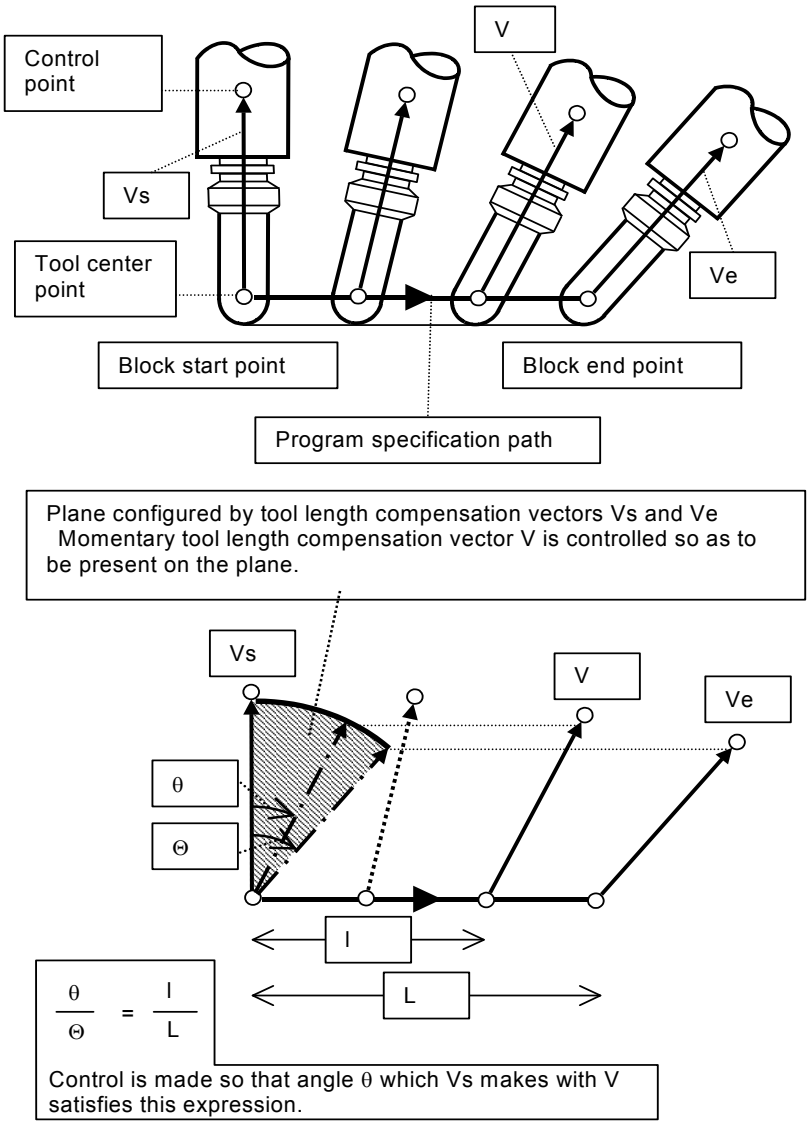


Fig. 22.4 (e) Tool posture control (positioning or linear interpolation)

In tool posture control (circular or helical interpolation), the two rotation axes are controlled so that the momentary tool posture (tool length compensation vector) satisfies the following conditions (Fig. 22.4 (e)).

Here, three directions are defined as follows:

- Tool direction : Direction from the tool center to the control point (tool posture)
- Center direction : Direction from the tool center to the center of the arc
- Traveling direction : Direction perpendicular to the center direction on the interpolation plane and in which the tool travels

The angles formed by the tool direction and other directions are defined as follows:

- α_s : Angle formed by the tool direction and the center direction at the block start point
- β_s : Angle formed by the tool direction and the traveling direction at the block start point
- α_e : Angle formed by the tool direction and the center direction at the block end point
- β_e : Angle formed by the tool direction and the traveling direction at the block end point
- α_t : Angle formed by the momentary tool direction and the center direction

- β_t : Angle formed by the momentary tool direction and the traveling direction

If, during tool posture control, circular or helical interpolation is specified, the momentary tool direction is controlled to the direction determined with α_t, β_t that satisfy the expressions below. (Fig. 22.4 (f)).

$$\begin{cases} \alpha_t = \alpha_s + (\alpha_e - \alpha_s) \cdot t \\ \beta_t = \beta_s + (\beta_e - \beta_s) \cdot t \end{cases}$$

where t represents the rate of movement of the tool center (rate of the momentary movement amount in relation to the total movement of that block of the tool center). $t = 0$ at the block start point and $t = 1$ at the block end point.

Thus, to move the tool direction (tool posture) within the side of the cone, they are usually specified so that $\alpha_s = \alpha_e, \beta_s = \beta_e = 90\,\mathrm{deg}$. Even if they are not specified this way, the tool direction is controlled according to the above algorithm.

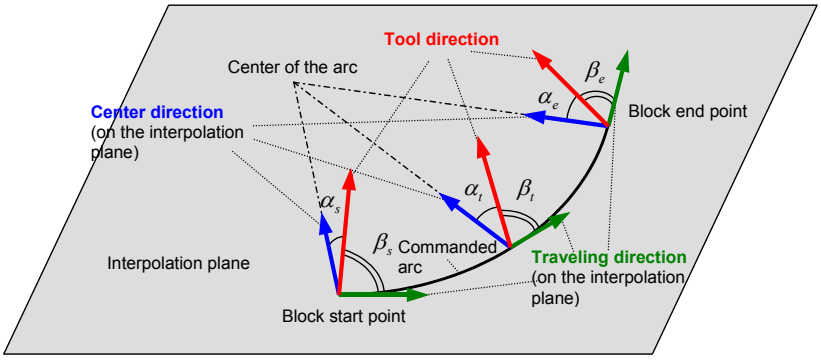


Fig. 22.4 (f) Method of tool posture control (circular interpolation)

Other notes and restrictions are the same as those for tool center point control.

Singular point

- "Tool-side rotary axis" and "workpiece-side rotary axis"

When tool center point control type 2 is specified, programming needs to be performed considering a singular point and singular point posture. The "tool-side rotary axis" and "workpiece-side rotary axis" used in the descriptions of a singular point and singular point posture later are explained below.
On a 5-axis machine that has two rotary axes, one rotary axis has a function for tilting the tool toward a workpiece. This rotary axis is referred to as a "tool-side rotary axis".
The other rotary axis is referred to as a "workpiece-side rotary axis".
The table below indicates the axes that represent these rotary axes for each mechanical unit type.

Table 22.4 (a) "Tool-side rotary axis" and "workpiece-side rotary axis"

Mechanical unit type (parameter No. 19680)	Tool-side rotary axis	Workpiece-side rotary axis
Tool rotation type (2)	Slave axis	Master axis
Table rotation type (12)	Master axis	Slave axis
Composite type (21)	Tool rotary axis	Table rotary axis

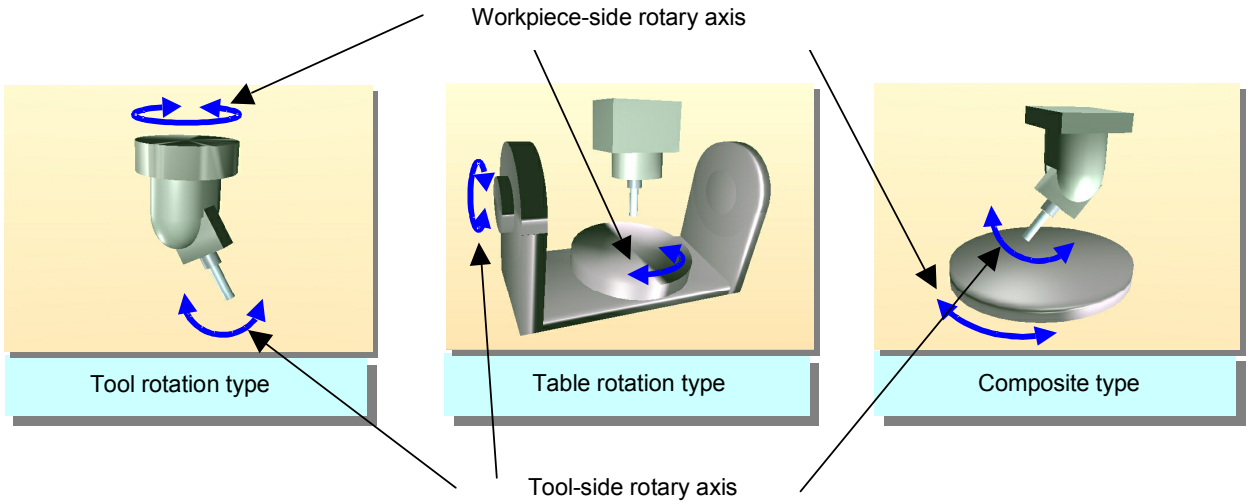


Fig. 22.4 (g) "Tool-side rotary axis" and "workpiece-side rotary axis"

- Singular point, singular point posture

A tool posture is uniquely determined when the angular displacements of the two rotary axes are determined. However, a combination of the angular displacements of the two rotary axes for determining a particular tool posture is not uniquely determined usually.

In particular, a tool posture with the angular displacement of the workpiece-side rotary axis being arbitrary is referred to as a "singular point posture", and the angular displacement of the tool-side rotary axis that causes the tool posture to assume a singular point posture is referred to as the "singular point (or singular point angular displacement)". When the angular displacement of the tool-side rotary axis is at a singular point, the center of the workpiece-side rotary axis and the tool posture (tool orientation) are parallel with each other.

Example)

Suppose that a tool rotation type machine is used, the master axis is the C-axis (about the Z-axis), the slave axis is the B-axis (about the Y-axis), and the reference tool axis direction is the Z-axis direction. In this case, singular point B = ..., 0°, 180°, ..., and a singular point posture (the same direction as the Z-axis direction or the direction opposite to the Z-axis direction) results at an arbitrary C-axis angular displacement.

For example, when the B-axis angular displacement is B = 0 (singular point) as shown in Fig. 22.4 (h), the same tool posture (singular point posture) is assumed at an arbitrary C-axis angular displacement.

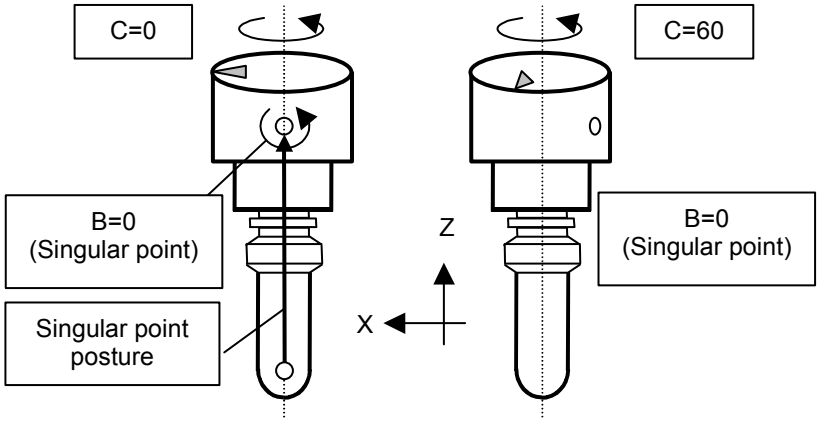
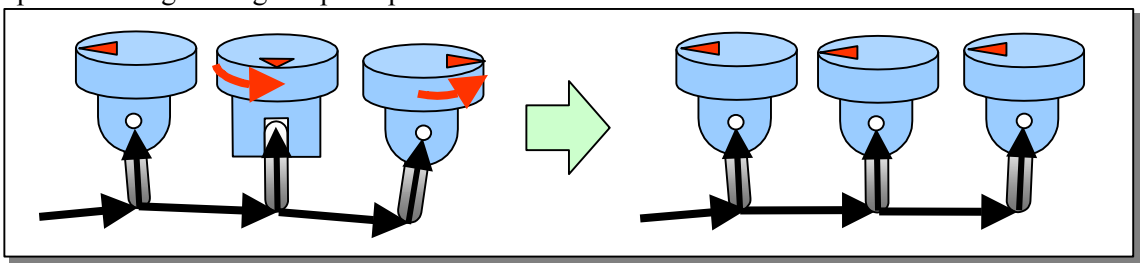


Fig. 22.4 (h) Singular point and singular point posture

There is no singular point and singular point posture on some types of machines as in the case where the reference tool axis direction is tilted (parameter No. 19698 and No. 19699) or a rotary axis is an angular axis (parameter No. 19682, No. 19683, No. 19687, and No. 19688).

- When the tool posture is close to a singular point posture

When tool posture control is exercised on a machine that has a singular point, and the tool posture becomes close to a singular point posture during execution of a block, the operation of the workpiece-side rotary axis can become so large, resulting in unstable machine operation.
In such a case, machine operation can be made stabler, the machining time can be reduced, and the surface roughness can be improved by changing the tool posture at the block end point so that the tool posture passes through a singular point posture.



A large and abrupt movement is made on the workpiece-side rotary axis when the tool posture passes through a singular point close posture.

A large and abrupt movement on the workpiece-side rotary axis can be avoided by passing through a singular point posture. By passing through a singular point posture, the tool posture at the block end is changed within the allowable range.

Fig. 22.4 (i) Changing the tool posture at a block end point

When tool posture control is used, and the angle between a tool posture and singular point posture is equal to or less than the value of (parameter No. 19738), the tool posture is determined to be close to a singular point posture. The tool posture at this time is referred to as a "singular point close posture". In positioning or linear interpolation, if tool center point control is specified (G43.5) with type 2, which specifies the tool posture at the block end point with a tool axis direction (I, J, K), the tool posture is checked in each block to see if it becomes a singular point close posture. If it becomes a singular point close posture, the tool posture at that block end point is changed (the rotation axis angle is changed) so that the posture passes through the singular point posture.
If a singular point close posture is assumed in a block, the tool posture at the end point of the block is changed (the rotary axis angular displacement is changed) to pass through a singular point posture. If the end points before and after the workpiece-side rotary axis is changed vary largely from each other as the result of changing the tool posture at the block end point, the change in the tool posture at the block end point can be suppressed.
However, if the value of parameter No. 19738 is 0, the tool posture at the block end point is not changed.
If the tool posture at the start point or end point of a block is a singular point posture or the tool posture becomes a singular point posture in the middle of a block, the tool posture is not changed at the block end point.

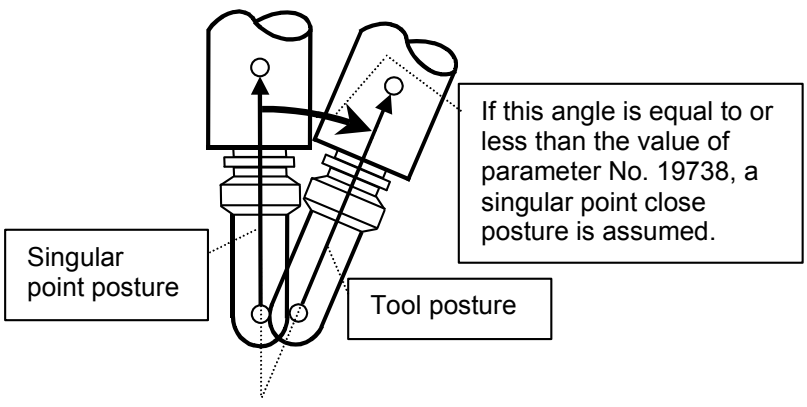


Fig. 22.4 (j) Singular point close posture

If a singular point close posture is assumed during execution of a block, change the angular displacements of the rotary axes at the end point of the block calculated from the tool posture specified by (I, J, K) so that the tool posture passes through a singular point posture in the block as indicated below.

Tool-side rotary axis:

Turned reversely relative to the singular point angular displacement

Example:

When the end point angular displacement before a change is 60°, and the singular point angular displacement is 20°, the end point angular displacement after a change is -20°.

Workpiece-side rotary axis:

Same as the angular displacement at the start point

Example:

When the end point angular displacement after a change is 30°, and the start point angular displacement is -150°, the end point angular displacement after a change is -150°.

However, if both of the difference between the end point angular displacement before a change and (end point angular displacement after a change + 180°) and the difference between the end point angular displacement before a change and (end point angular displacement after a change - 180°) are equal to or greater than the value of parameter No. 19739, the program is executed according to the setting of bit 3 (NPC) of parameter No. 19696, suppressing the change, or alarm PS5421 is issued.

This function is used to avoid unstable machine operation in a block where a singular point close posture is assumed as shown in Fig. 22.4 (i).

Notes

- When the reference tool axis or a rotary axis is inclined

When tool center point control is used, the inclination of the reference tool axis can be set in parameter No. 19698 and No. 19699, and the inclination of a rotary axis can be set in parameter No. 19682, No. 19683, No. 19687 and No. 19688.

On a machine with the reference tool axis or a rotary axis being inclined, all tool postures cannot be assumed. Some tool postures can never be assumed by setting the angular displacements of the two rotary axes in any way. If the tool posture passes through any of those impossible tool postures during program execution when tool posture control is enabled, alarm PS0432 is issued before execution of the block.

Example:

On a tool rotation type machine where the master axis is the C-axis (about the Z-axis), the slave axis is the B-axis (about the Y-axis), and the reference tool axis direction is inclined relative to the positive Z-axis direction (a non-zero value is set in parameter No. 19698 and No. 19699), no tool posture can be assumed in the positive or negative direction of the Z-axis. So, if a block specifies such a tool posture during tool posture control, alarm PS0432 is issued before execution of the block.

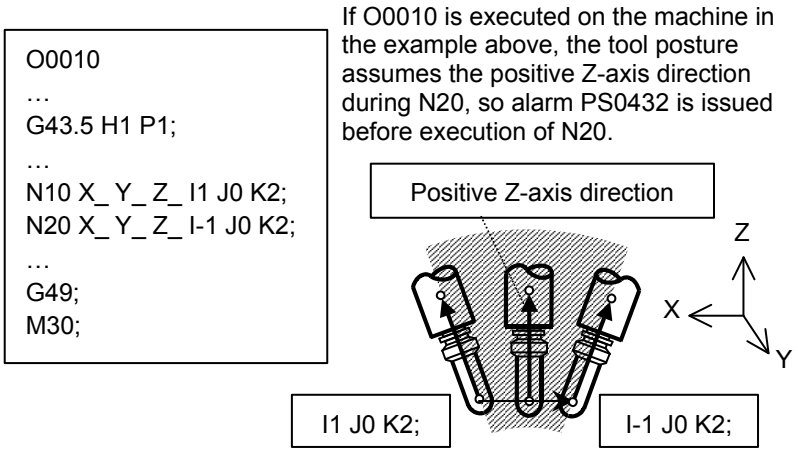


Fig. 22.4 (k) Example of impossible tool posture

- When the operation range of rotary axes is specified

When tool center point control is used, the operation range of rotary axes can be set in parameter No. 19741 to No. 19744.

If either of the rotary axes exceeds the set operation range during program execution when tool posture control is enabled, alarm DS0029 is issued.
If tool posture control is disabled, rotary axis operation range specification based on parameter No. 19741 to No. 19744 is enabled only when the tool center point control type 2 command (G43.5) is specified. If tool posture control is enabled, the check mentioned above can be made even when the tool center point control type 1 command (G43.4) is specified.
If the operation range of the first rotary axis is not specified, set 0 in both of parameter No. 19741 and No. 19742.
If the operation range of the second rotary axis is not specified, set 0 in both of parameter No. 19743 and No. 19744.

- When tool center point control type 1 is used

Tool posture control can also be exercised by using the tool center point control type 1 command (G43.4), which specifies the tool posture at the end point of a block with the angular displacements of the two rotary axes. However, when tool posture control is exercised using the tool posture at the end point of a block calculated from the rotary axis angular displacement specified by the program, alarm PS5421 is issued before execution of the block if the program-specified rotary axis angular displacement does not result.
Example:

Suppose that a tool rotation type machine where the master axis is the C-axis (about the Z-axis), the slave axis is the B-axis (about the Y-axis), the reference tool axis direction is the Z-axis direction, and the reference tool axis and rotary axes are not inclined is used. If the rotary axis angular displacements at the end point of a block are specified as (B, C) = (-90°, -90°) (with the tool posture being in the positive Y-axis direction) when the rotary axis angular displacements at the start point of the block are (B, C) = (90°, 0°) (with the tool posture being in the positive X-axis direction), tool posture control needs to maintain the tool posture in parallel with the XY plane. In this state, however, the specified end point (B, C) = (-90°, -90°) cannot be reached, so that alarm PS5421 is issued.
If (B, C) = (90°, 90°) for representing the same tool posture in the positive Y-axis direction at the end point of a block is specified, tool posture control is possible.

```
O0020
…
G43.4 H1 P1;
…
N10 X_ Y_ Z_ B90.0 C0.0;
N20 X_ Y_ Z_ B-90.0 C-90.0;
…
G49;
M30;
```

```
O0021
…
G43.4 H1 P1;
…
N10 X_ Y_ Z_ B90.0 C0.0;
N20 X_ Y_ Z_ B90.0 C90.0;
…
G49;
M30;
```

At the N20 end point of both of the programs O0020 and O0021, the tool posture is in the positive Y-axis direction. However, the N20 end point (B=-90.0, C=-90.0) of O0020 cannot be reached from the N10 end point (B=90.0, C=0.0) by tool posture control. So, alarm PS5421 is issued with O0020. However, the N20 end point (B=90.0, C-90.0) of O0021 can be reached by tool posture control.

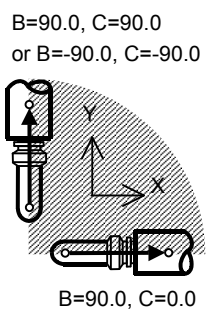


Fig. 22.4 (l) Example where tool posture control is impossible when type 1 is specified

- When the tool posture directions at the start point and end point of a block match each other

If, in positioning or linear interpolation, the tool posture directions at the start point and end point of a block match each other (either the same direction or the opposite direction), the plane made up of the tool postures at these points cannot be determined. So, tool posture control in that block is temporarily disabled. However, whether to enable/disable tool posture control is kept as modal information. So, tool posture control temporarily disabled for the cause mentioned above is enabled again in the subsequent blocks.

- Commands that change the direction of the tool in relation to the interpolation plane

In circular interpolation or helical interpolation, a command that changes the tool direction cannot be executed to the interpolation plane specified with G17, G18, or G19. If an attempt is made to execute such a command, alarm PS0432, "UNAVAILABLE POSTURE IN TPC", is issued.
Example

```
O0020
…
N10 G43.5 H1 P1;
…
N20 X_ Y_ I1 J0 K2;
N30 G17 G02 X_ Y_ R_ I1 J0 K-2;
…
```

In the program O0020 on the left, the tool direction is changed in relation to the interpolation plane (G17: XY plane) in N30. Thus, alarm PS0432 is issued.

Fig. 22.4 (m) Example of a command that changes the tool direction

- 3-dimensional circular interpolation

This function cannot be used with 3-dimensional circular interpolation.

For other notes, those described for tool center point control are applicable.

Limitation

Manual intervention cannot be performed during tool posture control.
If manual intervention is performed, alarm PS5421, "ILLEGAL COMMAND IN G43.4/G43.5", is issued at an operation restart.

For other restrictions, those described for tool center point control are applicable.

22.5 CUTTING POINT COMMAND

Overview

While the operation of the tool tip center is specified with tool center point control, the operation of the cutting point can be specified with the cutting point command. With this function, a corner-R tool can also be used.

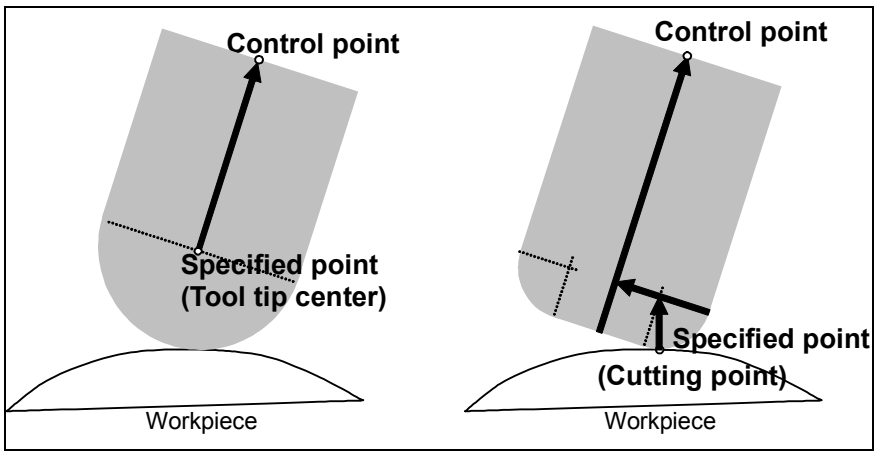


Fig. 22.5 (a) Tool center point control (left) and cutting point command (right)

NOTE

1 The following options are needed:
Machining Center System:
- Cutting point command (Note 2)
- Tool center point control (Note 2)

Lathe System:
- Cutting point command (Note 2)
- Tool center point control (Note 2)
- Tool offset for milling and turning function
- Tool geometry/wear compensation

Multiple system:
- Cutting point command (Note 2)
- Tool center point control (Note 2)
- Tool offset for milling and turning function
- Tool offset memory C
- Tool geometry/wear compensation

2 The option of tool center point control and cutting point command are included in the option of high-speed smooth TCP.

Format

There are two types, as described below, one of which is used depending on how the direction of the tool axis is specified.

(1) Type 1
The block end position of the rotation axis is specified (e.g. A, B, C).
The CNC performs tool length compensation by the specified compensation amount in the tool axis direction that is calculated from the specified position of the rotation axis and controls so that the tip of the tool moves along the specified path.

(2) Type 2
The direction of the tool axis (I, J, K) at the block end point, as seen from the coordinate system fixed on the table, is specified, instead of the position of the rotation axis.
The CNC calculates the end position of the rotation axis so that the tool will face the specified direction at the end point, performs tool length compensation by the compensation amount in the tool axis direction that is calculated from the position of the rotation axis, and controls so that the tip of the tool moves along the specified path.

M

```
G43.8 IP_ α_ β_ H_ (D_) ,L2 I_ J_ K_;
                          Turn cutting point command (type 1) mode ON
IP_ α_ β_ ,L2 I_ J_ K_;
 :                        Cutting point command mode
 :
G49 ;                     Turn cutting point command mode OFF

G43.9 IP_ H_ (D_) ,L2 I_ J_ K _;
                          Turn cutting point command (type 2) mode ON
IP_ I_ J_ K_ ,L2 I_ J_ K _;
 :                        Cutting point command mode
 :
G49 ;                     Turn cutting point command mode OFF
```

IP	:	In the case of an absolute command, the coordinates of the cutting point at the block end point In the case of an incremental command, the amount of movement of the cutting point
α, β	:	In the case of an absolute command, the coordinates of the rotation axis at the block end point In the case of an incremental command, the amount of movement along the rotation axis
I, J, K	:	Direction of the tool axis at the block end point, as seen from the coordinate system fixed on the table
H, D	:	Tool offset number (tool length, radius and corner-R amount) The specification method is different depending on the kind of the tool compensation memory.
I, J, K following ",L2"	:	Direction perpendicular to the cutting surface at the block end point, as seen from the coordinate system fixed on the table. (Specify only I, J, or K after ",L2".)

Tool length, radius and corner-R compensation value is set to the offset memory corresponding to an tool offset number beforehand.
To the setting and the display of the compensation value, the specification of each tool compensation memory is applied in machining center system,
Please use tool compensation memory C usually. In case of tool compensation memory C, each compensation value can be set to the same offset number because there is a compensation memory in each tool length / radius / corner R. Therefore, the setting and the confirmation of each compensation value are easy.

In case of tool compensation memory A and B, each compensation value is set to a different offset number becuse there is no compensation memory in each tool length / radius / corner R.

Table22.5 (a) Specification format of tool offset number (machining center system)

		Tool offset memory C		Tool offset memory A and B	
		ON1 (No.11269#3)=1	ON1 (No.11269#3)=0	(No.11419)=0	(No.11419)=n
Specification format of tool offset number	Cutting point command (type 1)	G43.8 H_;	G43.8 H_ D_;	G43.8 H_;	G43.8 H_;
	Cutting point command (type 2)	G43.9 H_;	G43.9 H_ D_;	G43.9 H_;	G43.9 H_;
Tool offset number	Tool length	H code	H code	H code	H code
	Tool radius	H code	D code	H code + 1	H code + n
	Corner R	H code	D code	H code + 2	H code + 2 * n

(1) In case of tool offset memory C when bit 3 (ON1) of parameter No. 11269 is 1:
Tool offset number is specified only by H code. The tool offset number of each compensation value is a number specified by H code.
And, if D code is specified in G43.8/G43.9 block, alarm PS5464, "ILLEGAL COMMAND IN G43.8/G43.9", occurs.
Example)
G43.8 H10;
Tool length compensation value Tool offset number 10
Tool radius compensation value Tool offset number 10
Corner-R compensation value Tool offset number 10
Example)
G43.8 H10D20;
Alarm PS5464

(2) In case of tool offset memory C when bit 3 (ON1) of parameter No. 11269 is 0:
Tool offset number is specified by H/D code. The tool offset number of each compensation value is a number specified by H/D code as follows.
Example)
G43.8 H10D20;
Tool length compensation value Tool offset number 10
Tool radius compensation value Tool offset number 20
Corner-R compensation value Tool offset number 20

(3) In case of tool offset memory A and B:
Tool offset number is specified only by H code. The tool offset number of each compensation value is automatically assigned by NC which assigns the number specified by H code as a head. The interval of the offset number can be set according to parameter No. 11419.
When [H code + 2] or [H code + 2 * (parameter No. 11419)] is larger than the maximum number of tool compensation values, alarm PS0030, "ILLEGAL OFFSET NUMBER", occurs.
And, if D code is specified in G43.8/G43.9 block, alarm PS5464, "ILLEGAL COMMAND IN G43.8/G43.9", occurs.
Example)
G43.8 H10; (when parameter No. 11419 is 0)
Tool length compensation value Tool offset number 10
Tool radius compensation value Tool offset number 11
Corner-R compensation value Tool offset number 12

G43.8 H10; (when parameter No. 11419 is 10)

Tool length compensation value	Tool offset number 10
Tool radius compensation value	Tool offset number 20
Corner-R compensation value	Tool offset number 30

Example)
G43.8 H10D20;
Alarm PS5464

T

G43.8 IP_ α_ β_ D_ ,L2 I_ J_ K_;
Turn cutting point command (type 1) mode ON
IP_ α_ β_ ,L2 I_ J_ K_;
: **Cutting point command mode**
:
G49 ; **Turn cutting point command mode OFF**

G43.9 IP_ D_ ,L2 I_ J_ K _;
Turn cutting point command (type 2) mode ON
IP_ I_ J_ K_ ,L2 I_ J_ K _;
: **Cutting point command mode**
:
G49 ; **Turn cutting point command mode OFF**

IP : In the case of an absolute command, the coordinates of the cutting point at the block end point
In the case of an incremental command, the amount of movement of the cutting point
α, β : In the case of an absolute command, the coordinates of the rotation axis at the block end point
In the case of an incremental command, the amount of movement along the rotation axis
I, J, K : Direction of the tool axis at the block end point, as seen from the coordinate system fixed on the table
D : Tool offset number (tool length, radius and corner-R amount)
I, J, K following ",L2" :
Direction perpendicular to the cutting surface at the block end point, as seen from the coordinate system fixed on the table. (Specify only I, J, or K after ",L2".)

Tool length, radius and corner-R compensation value is set to the offset memory corresponding to an tool offset number beforehand.
To the setting and the display of the compensation value, the specification of tool offset for milling and turning function is applied in lathe system.
Each compensation value can be set to the same offset number because there is a compensation memory in each tool length / radius / corner R.

Table22.5 (b) Specification format of tool offset number (lathe system)

Specification format of tool offset number	**Cutting point command (type 1)**	G43.8 D_;
	Cutting point command (type 2)	G43.9 D_;
Tool offset number	**Tool length**	D code
	Tool radius	D code
	Corner R	D code

Tool offset number is specified only by D code. The tool offset number of each compensation value is a number specified by H/D code as follows.
Example)

G43.8 D10;

Tool length compensation value Tool offset number 10
Tool radius compensation value Tool offset number 10
Corner-R compensation value Tool offset number 10

In cutting point command mode, the control point is calculated for each of a ball-end mill, square-end mill, and radius-end mill (tool with corner-R) from the specified cutting point, tool length compensation specified by H, and tool radius compensation and corner-R offset specified by D, as described below, and the tool moves with the calculated control point.
When the corner-R offset is equal to the tool radius compensation, if the ball-end mill and the corner-R offset are set to 0, the tool is regarded as a square-end mill.

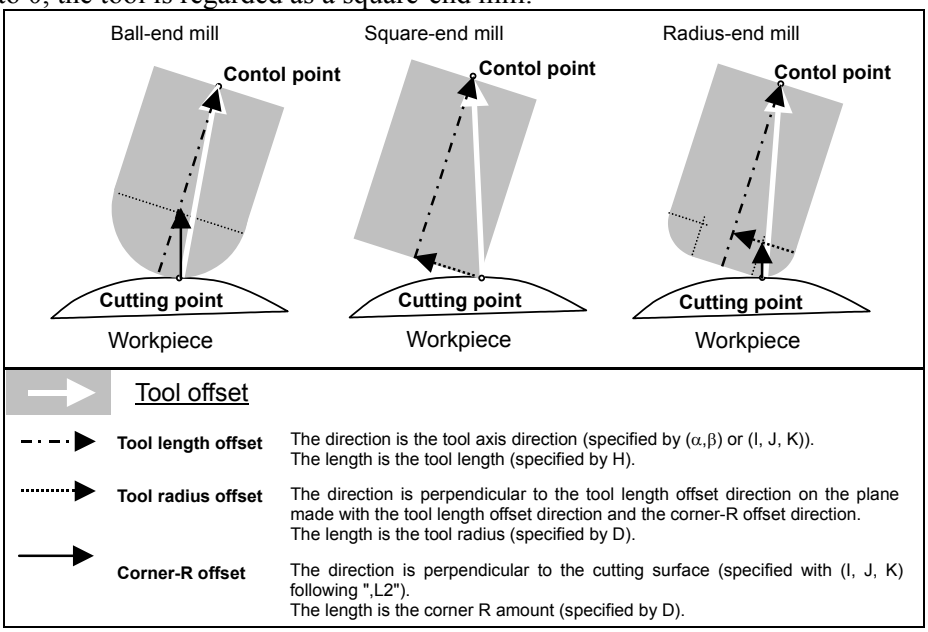


Fig. 22.5 (b) Tool offset

In a block of G43.8/G43.9, if a part of (I, J, K) following “,L2” is omitted, the omitted value is regarded as 0, and if all are omitted, the tool direction is regarded as the same as the reference tool direction, specified by parameters Nos. 19697, 19698, and 19699. If, after the block of G43.8/G43.9, a part of (I, J, K) following “,L2” is omitted, the omitted value is regarded as 0, and if all are omitted, the direction is specified by (I, J, K) following ",L2" in the previous block. If all of (I, J, K) following “,L2” are specified as 0, alarm PS5464 occurs.
The number of effective digits after the decimal point of (I, J, K) following “,L2” is the same as the increment system.

Example)

N10 G90 G00 X0.0 Y0.0 Z0.0 A0.0 C0.0 ; N20 G43.8H1 D1 ,L2 I-1.0 J0.0 K10.0 ; N30 G01 X20.0 Y0.0 Z0.0 A0.0 C0.0 F1800.0,L2 I-1.0 J-1.0 K10.0 ; N40 G01 X20.0 Y20.0 Z0.0 A10.0 C5.0 ; N50 G01 X0.0 Y20.0 Z0.0 A20.0 C7.0 ,L2 I0.0 J-1.0 K1.0 ; ….. N200 G49 ;	Move to the starting point. Turn cutting point command mode ON. Specify an F command before “,L2". For (I, J, K) following “,L2”, those in the previous block are inherited. Turn cutting point command mode OFF.

- Tool Posture Control

It is possible to use tool posture control together with the cutting point command. To use them together, specify P1 in a G43.8/G43.9 block or set bit 0 (TPC) of parameter No. 19604 to 1. For details of posture control, see Section 22.4, "TOOL POSTURE CONTROL".

Explanation

- Machine configuration

The concept about the machine configuration is the same as that of tool center point control. For details, see "Machine configuration" in Section 22.1, "TOOL CENTER POINT CONTROL".

- Singular posture

In the case of the square-end mill and the radius-end mill, when the tool direction is perpendicular to the cutting surface, the tool can be said in an unstable state because, although the tool contacts the cutting surface at a single contact point, the coordinate on the linear axis cannot be determined to be a single one. Such a tool posture is called a singular posture. With the cutting point command, the tool in a singular posture can be prevented from being unstable by setting parameter No. 11262. Set an angle (0.0 to 90.0 (deg)) in parameter No. 11262. The tool posture is considered to be a near-singular posture if the angle formed by the tool length and the direction perpendicular to the cutting surface is less than the setting of parameter No. 11262 (Fig. 22.5 (c)). No singular posture is regarded to exist if parameter No. 11262 is 0.0. If the posture is regarded to be a near-singular posture, the unstable state peculiar to a singular posture can be avoided by fixing the "vector from the specified point to the tool center point" with the "vector from the cutting point to the tool center point" immediately before the posture is regarded as a near-singular posture. At this time, the point shifted from the specified point becomes the cutting point (Fig. 22.5 (d)). Do not set too large a value in parameter No. 11262 because the larger the value of parameter No. 11262, the larger the shift.

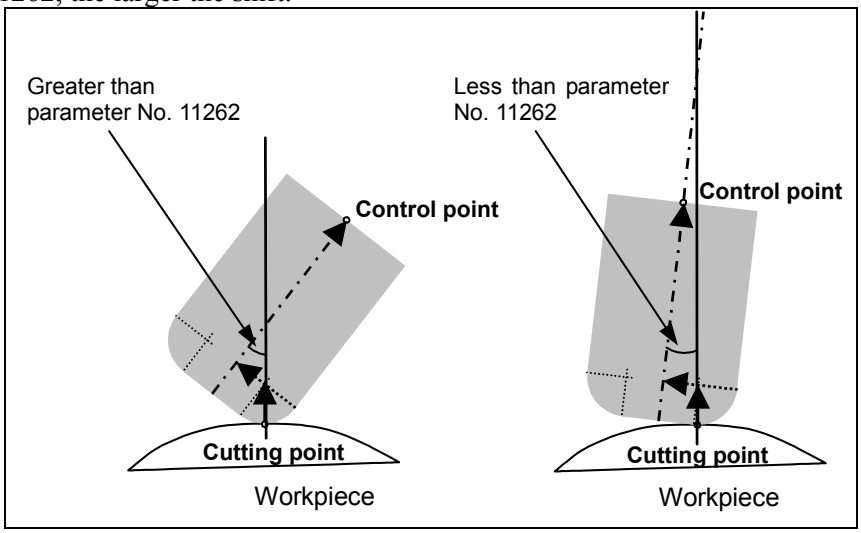


Fig. 22.5 (c) Near-singular posture (right)

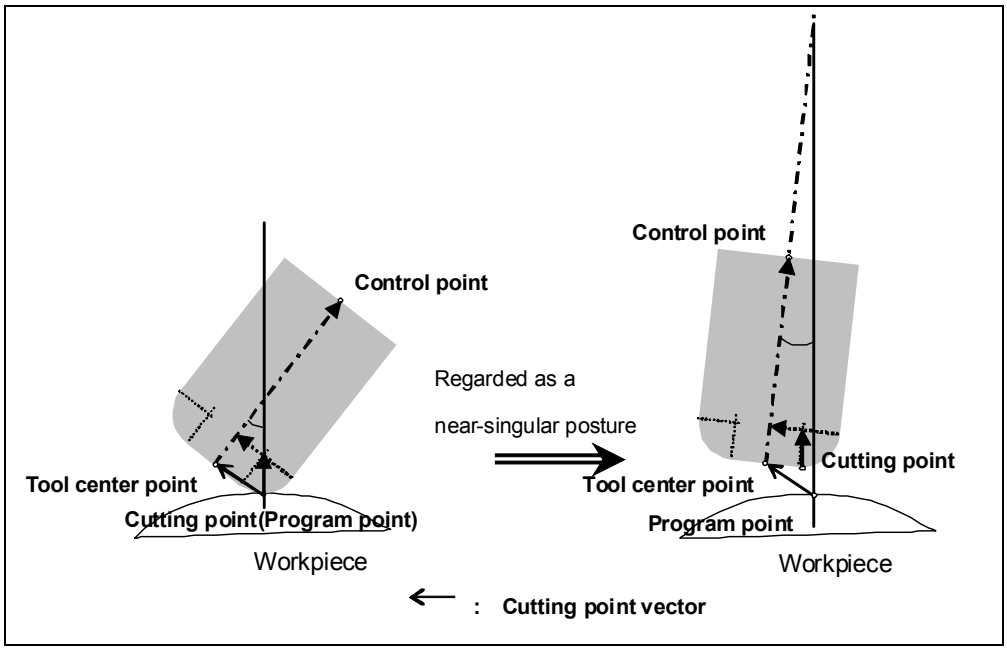


Fig. 22.5 (d) If the posture is a near-singular posture, the "vector from the specified point to the tool center point" is fixed with the "vector from the cutting point to the tool center point" immediately before the posture is regarded as a near-singular posture

- Commands that change the direction of the tool in relation to the interpolation plane

In circular interpolation, a command that changes the direction perpendicular to the cutting surface to the plane specified with G17, G18, or G19 in circular interpolation cannot be executed. If an attempt is made to execute such a command, alarm PS0432, "UNAVAILABLE POSTURE IN TPC", is issued.
Example)

O0020 … N10 G43.8 H1; … N20 X_ Y_ ,L2 I1 J0 K2; N30 G17 G02 X_ Y_ R_ ,L2 I1 J0 K-2; …	In the program O0020 on the left, the direction perpendicular to the cutting surface is changed in relation to the interpolation plane (G17: XY plane) in N30. Thus, alarm PS0432 is issued.

Fig. 22.5 (e) Example of a command that changes the direction perpendicular to the cutting surface

- Wrong operation prevention functions

By using the wrong operation prevention functions, it is possible to check whether data is in the valid range or not when the tool offset data is set.
In machining center system (tool offset memory C), and lathe system, the valid range set to "RADIUS" is also applied to corner-R compensation value.
Please refer to the chapter of wrong operation prevention functions for details of the function.

Restrictions

- Manual intervention

Manual intervention cannot be performed on the table rotation axis.
In the case of a tool rotation type machine configuration, if automatic operation is stopped and manual intervention is performed on the rotation axis, alarm PS5464 occurs.

- Acceleration/deceleration before look ahead interpolation

When the system is placed in cutting point command mode, acceleration/deceleration before look ahead interpolation is automatically enabled. Set the parameters for acceleration/deceleration before look ahead interpolation. If any setting is invalid, alarm PS5420 occurs.

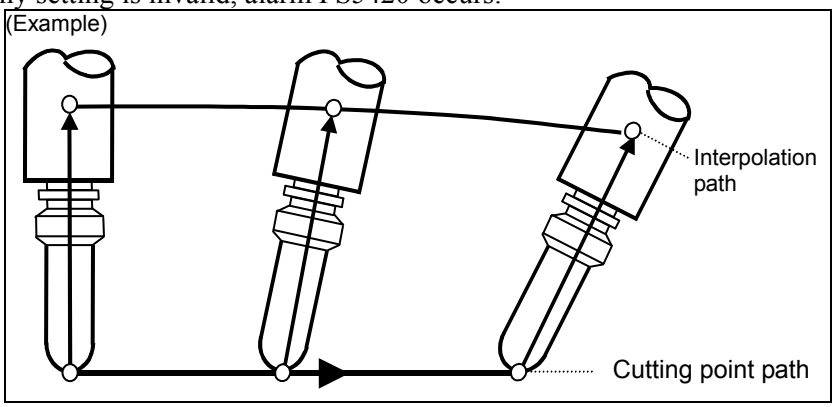


Fig. 22.5 (f)

In the case of the example in Fig. 22.5 (f), the feedrate is controlled so that the specified feedrate is attained at the cutting point and, therefore, the feedrate on the interpolation path is greater than the specified feedrate. At this time, the feedrate is clamped so that the feedrate on the interpolation path does not exceed the maximum cutting feedrate or the rapid traverse rate.

- Deceleration at a corner

While the cutting point command is in progress, the control point may operate with a curve even if the command is a linear one. Depending on the command, the control point may operate with a corner.
If AI contour control is not used, the machine may get a shock at a corner.
Thus, it is recommended to use AI contour control together to set the permissible feedrate difference (parameter No. 1783) and the permissible acceleration (parameters Nos. 1660 and 1737) at a corner.

- Hypothetical axis

The cutting point command cannot be used on a machine on which a hypothetical axis is set. Do not use the cutting point command on a machine on which a hypothetical axis is set.

- Stored limit check before move

While the cutting point command is in progress, stored limit check before move is disabled.

- Type 2

If, on a machine whose axis configuration is table rotation type or composite type (parameter No. 19680 = 12 or 21), a type 2 command (G43.5 command) is to be executed, it is necessary to make the setting for using the table coordinate system as the programming coordinate system (bit 5 (WKP) of parameter No. 19696 = 0). If an attempt is made to execute a type 2 command if the setting is that for using the workpiece coordinate system as the programming coordinate system, alarm PS5459 is issued.

- Unavailable operations

While the cutting point command is in progress, the operations below cannot be performed. While the cutting point command is in progress, do not perform these operations.

- Machine lock
- High-speed program check
- Program restart
- Tool retract and recover
- Retrace
- Active block cancel
- Manual handle retrace

- Functions that cannot be used together

While the cutting point command is in progress, the following functions cannot be used. While the cutting point command is in progress, do not use these functions.

- Parallel axis control
- Twin table control

- Specifiable G codes

The G codes that can be specified in cutting point command mode are as follows.
Do not specify G codes other than these.

- Positioning (G00)
- Linear interpolation (G01)
- Circular interpolation (G02,G03)
- Tool length compensation cancel (G49,G49.1)
- Absolute programming (G90)
- Incremental programming (G91)

- Modal G code states in which the cutting point command is specified

The cutting point command can be specified in the modal G code states below.
Do not specify the cutting point command in modal states other than these.

- Modal G codes included in "Specifiable G codes" above
- Polar coordinate interpolation mode cancel (G13.1,G113)
- Polar coordinate command mode cancel (G15)
- Tool offset cancel (G40)
- Normal direction control cancel (G40.1)
- Scaling cancel (G50)
- Programmable mirror image cancel (G50.1)
- Polygon turning cancel (G50.2)
- Rotary table dynamic fixture offset cancel (G54.2P0)
- Workpiece setting error compensation cancel (G54.4P0)
- Cutting mode (G64)
- Macro modal code cancel (G67)
- Coordinate system rotation / 3-dimensional coordinate system conversion cancel (G69)
- Canned cycle cancel (G80)
- Feed per minute (G94)
- Constant surface speed control cancel (G97)

- Specification of axes not related to the cutting point command

Axes not related to the cutting point command cannot be specified. While the cutting point command is in progress, do not specify any axes not related to the cutting point command.

- Linear axes of the cutting point command

The basic three axes set in parameter No. 1022 are assumed to be the three linear axes of the cutting point command. Axes parallel to the basic three axes cannot be used as the linear axes of the cutting point command.
If the basic three axes are not set in parameter No. 1022, alarm PS5459 occurs.

T

- Extended tool selection

When cutting point command is used with a lathe system, the tool change method needs to be ATC type (bit 3 (TCT) of parameter No. 5040 = 1).

In this case, only the value set in "Z/LENGTH", "NOSE R/RAD", and "CORNER R" on the tool compensation memory screen is used as a tool length, radius, and corner-R compensation value. Set values such as "X", "Y" axis offset values and so on are not used.

22.6 TILTED WORKING PLANE INDEXING

22.6.1 Tilted Working Plane Indexing

Overview

Programming for creating holes, pockets, and other figures in a datum plane tilted with respect to the workpiece would be easy if commands can be specified in a coordinate system fixed to this plane (called a feature coordinate system). This function enables commands to be specified in the feature coordinate system. The feature coordinate system is defined in the workpiece coordinate system.
For explanations about the relationship between the feature coordinate system and workpiece coordinate system, see Fig. 22.6.1 (a).

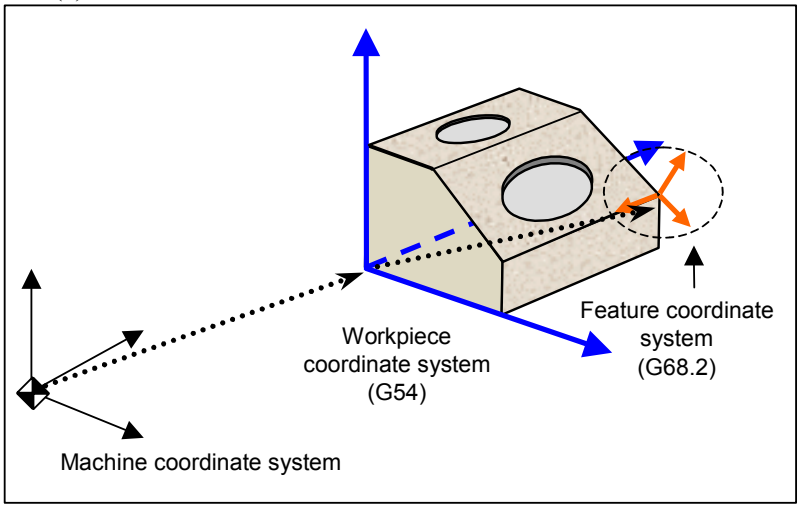


Fig. 22.6.1 (a) Feature coordinate system

The G68.2 command causes the programming coordinate system to switch to the feature coordinate system. The commands in all subsequent blocks are assumed to be specified in the feature coordinate system until G69 appears.
If G68.2 specifies the relationship between the feature coordinate system and the workpiece coordinate system, G53.1 automatically specifies the +Z direction of the feature coordinate system as the tool axis direction even if no angle is specified for the rotary axis. (See Fig. 22.6.1 (c).)
For explanations about the tool axis direction, see Fig. 22.6.1 (b).

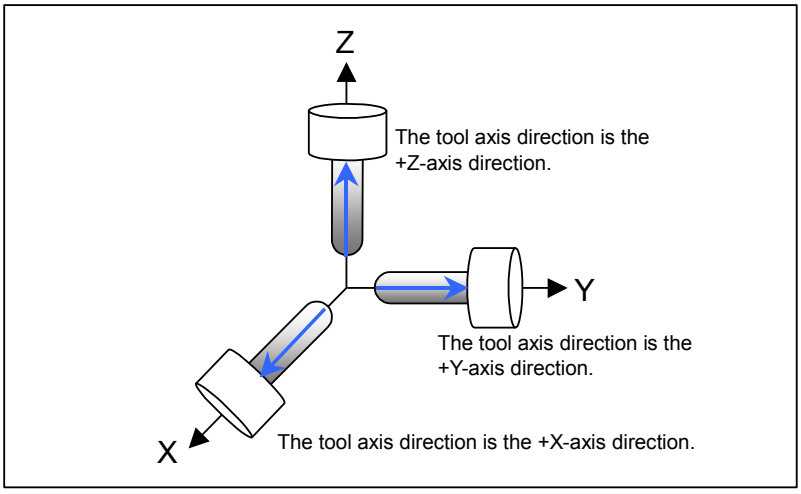


Fig. 22.6.1 (b) Tool axis direction

This function regards the direction normal to the machining plane as the +Z-axis direction of the feature coordinate system. After the G53.1 command, the tool is controlled so that it remains perpendicular to the machining plane.

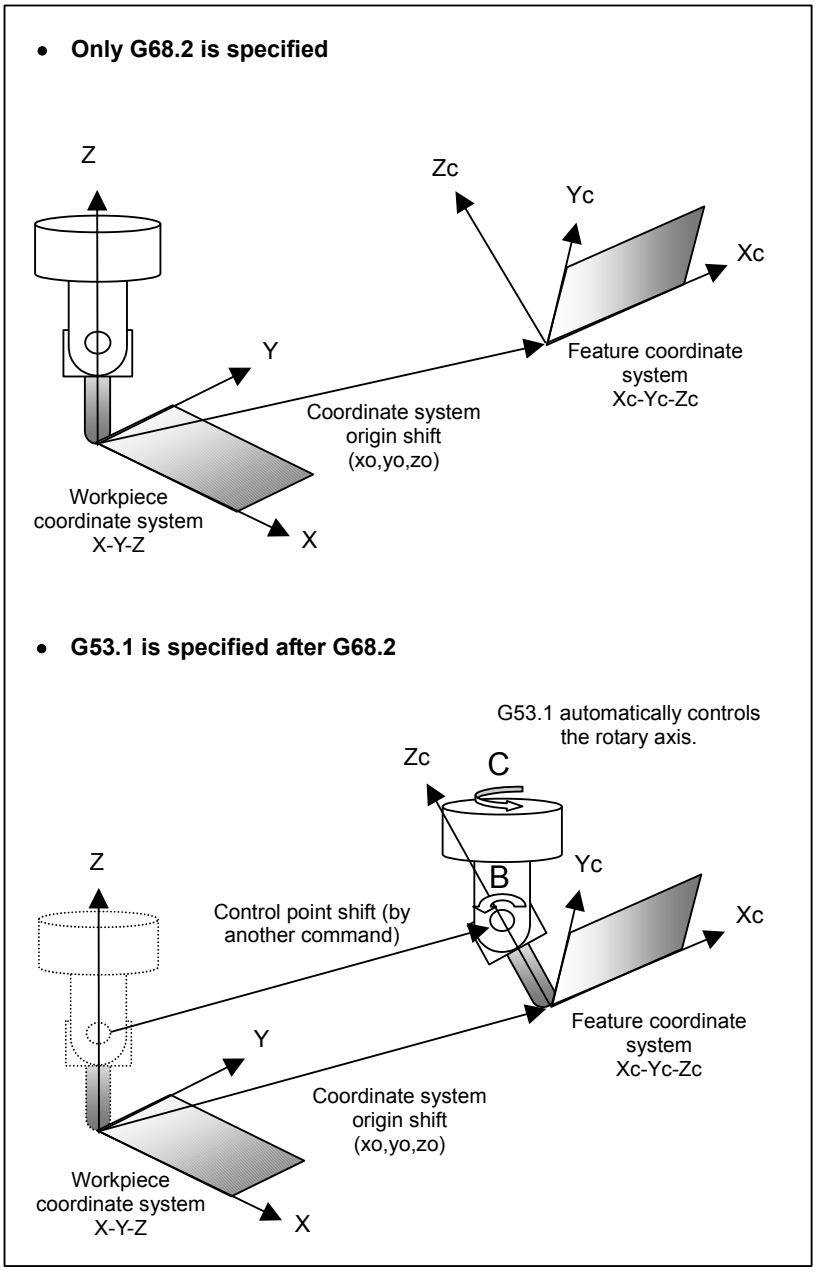


Fig. 22.6.1 (c) G68.2 and G53.1 commands

This function is applicable to the following machine configurations. (See Fig. 22.6.1 (d).)
<1> Tool rotation type machine controlled with two tool rotation axes
<2> Table rotation type machine controlled with two table rotation axes
<3> Composite type machine controlled with one tool rotation axis and one rotary axis

The function can also be used for a machine configuration in which the rotary axis for controlling the tool does not intersect the rotary axis for controlling the table.

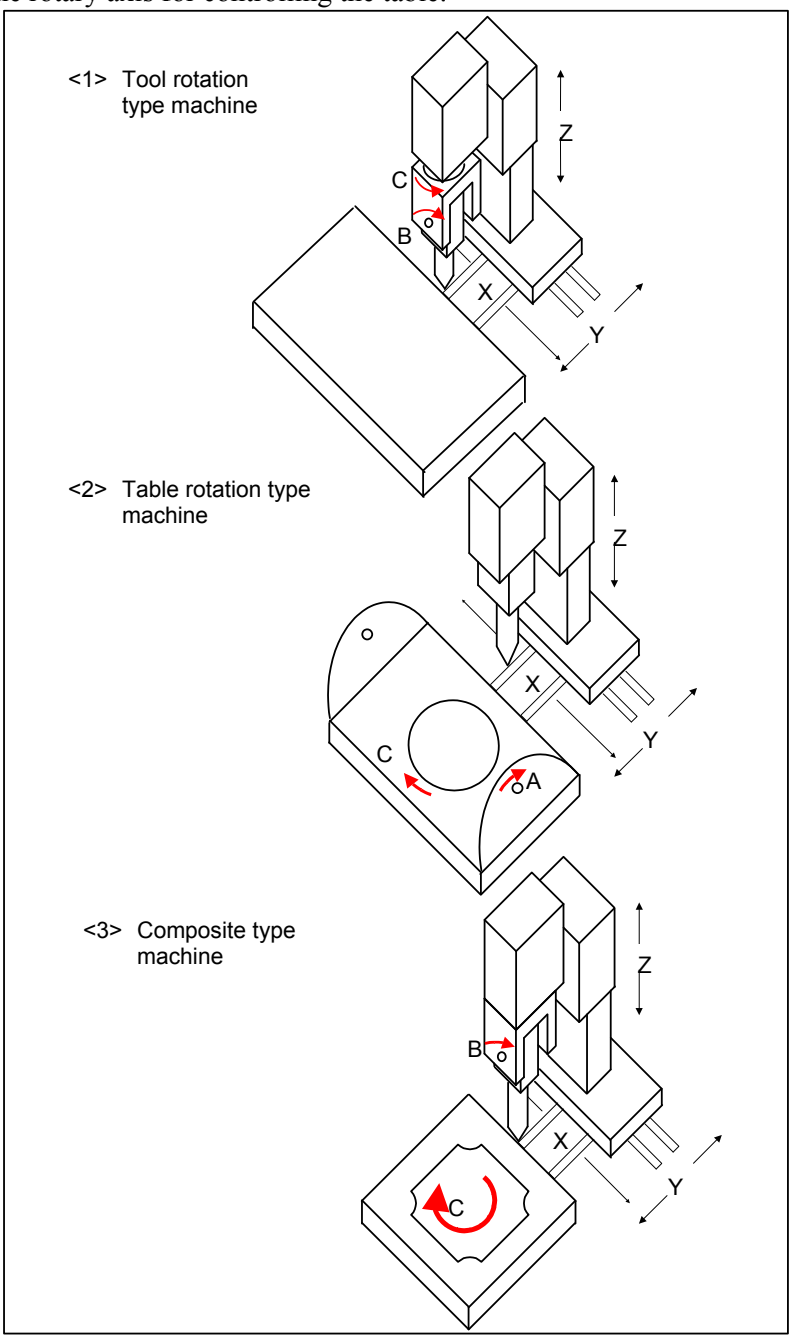


Fig. 22.6.1 (d) Three types of 5-axis machine

22.6.1.1 Tilted working plane indexing based on Eulerian angle

Format

- Tilted working plane indexing (G68.2)

M

G68.2 X $\underline{x}_0$ Y $\underline{y}_0$ Z $\underline{z}_0$ I$\underline{\alpha}$ J$\underline{\beta}$ K$\underline{\gamma}$;	Tilted working plane indexing
G69 ;	Cancels the tilted working plane indexing.

X,Y,Z: Feature coordinate system origin
The axes specified here are the three axes of the feature coordinate system. Specify the three basic axes or parallel axes set by parameter No. 1022. When specification is omitted, the X, Y, and Z of the basic three axes are assumed to be 0.
I,J,K : Euler's angle for determining the orientation of the feature coordinate system

T

G68.2 X $\underline{x}_0$ Y $\underline{y}_0$ Z $\underline{z}_0$ I$\underline{\alpha}$ J$\underline{\beta}$ K$\underline{\gamma}$;	Tilted working plane indexing
G69.1 ;	Cancels the tilted working plane indexing.

X,Y,Z: Feature coordinate system origin
The axes specified here are the three axes of the feature coordinate system. Specify the three basic axes or parallel axes set by parameter No. 1022. When specification is omitted, the X, Y, and Z of the basic three axes are assumed to be 0.
I,J,K : Euler's angle for determining the orientation of the feature coordinate system

- Tool axis direction control (G53.1)

G53.1 ;	Controls the tool axis direction.

⚠ CAUTION
1 G53.1 must be specified in a block after the block that contains G68.2. An alarm occurs if G53.1 is specified without G68.2 being specified in a preceding block.
2 G53.1 must be specified in a block in which there is no other command.
3 The rotary axis moves at the maximum rapid traverse federate in the case of rapid traverse and at the specified federate in the case of cutting feed.
4 If a movement range is set for a roll over axis (parameters Nos. 19741 to 19744 $\neq$ 0), the roll over function is disabled during tool axis direction control (G53.1).

Explanation

- Coordinate conversion using an Euler's angle

Coordinate conversion by rotation is assumed to be performed around the workpiece coordinate system origin.

Let the coordinate system obtained by rotating the workpiece coordinate system around the Z-axis by an angle of α degrees be coordinate system 1. Similarly, let the coordinate system obtained by rotating coordinate system 1 around the X'-axis by an angle of β be coordinate system 2. The feature coordinate system is the coordinate system obtained by shifting the coordinate system that is obtained by rotating coordinate system 2 around the Z"-axis through an angle of γ degrees from the workpiece coordinate system origin by (Xo, Yo, Zo).

Fig. 22.6.1 (e) shows the relationship between the workpiece coordinate system and the feature coordinate system.

The figure also gives examples of displacement on the X-Y plane.

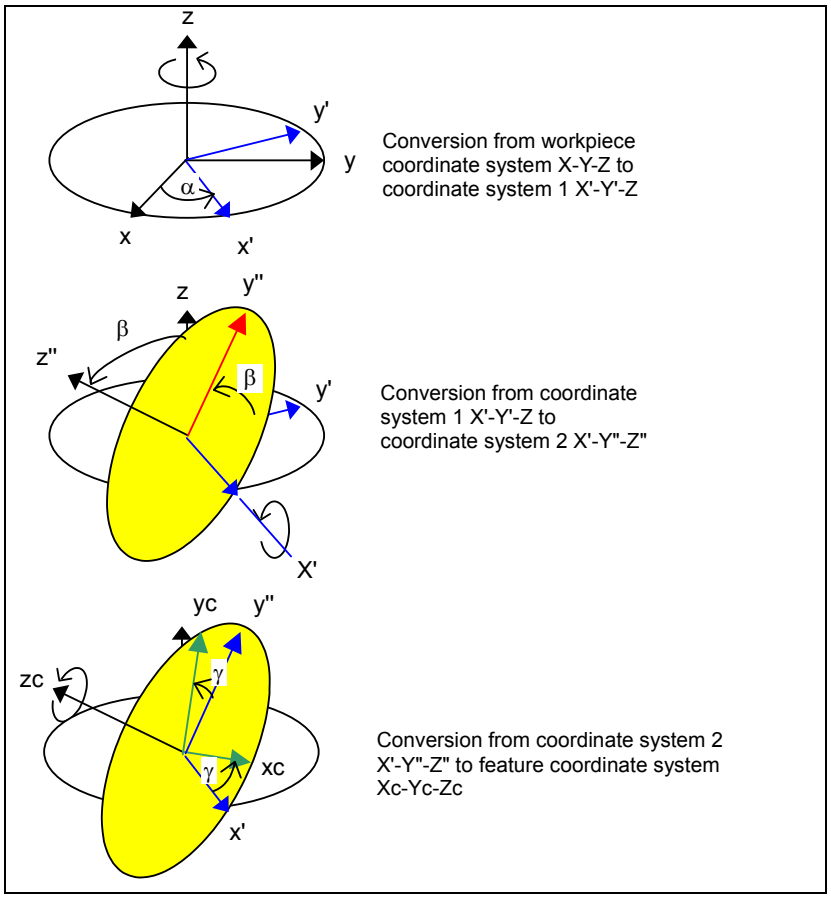


Fig. 22.6.1 (e) Coordinate conversion using an Euler's angle

- I0 J0 K0 command

When I0 J0 K0 is specified as an Euler's angle, the alarm (PS5457) (G68.2 format error) usually occurs. When bit 1 (ATW) of parameter No. 13451 is set to 1, the feature coordinate system with a tilted angle of 0 degree is used.

22.6.1.2 General specifications of the tilted working plane indexing

- Constant surface speed control

Constant surface speed control is exercised by using, as the reference, the machine axis specified in address P in a G96 block or the machine axis (not in the feature coordinate system but in the actually operating workpiece coordinate system) set in parameter No. 3770.

- Workpiece coordinate system selection command during the tilted working plane indexing

By executing the workpiece coordinate system selection command (G54 to G59, G54.1) during the tilted working plane indexing when bit 6 (3TW) of parameter No. 1205 is 1, it is possible to change the workpiece coordinate system. In this case, the coordinate system zero point shift of the tilted working plane indexing is maintained.

If an attempt is made to execute the workpiece coordinate system selection command (G54 to G59, G54.1) during the tilted working plane indexing when bit 6 (3TW) of parameter No. 1205 is 0, alarm PS5462 is issued.

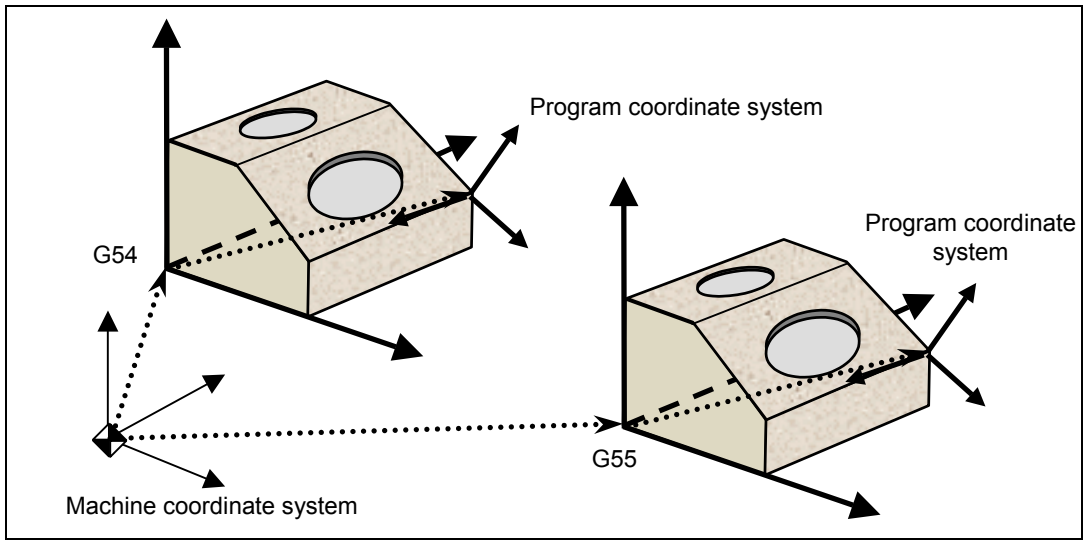


Fig. 22.6.1 (f)

・Minimum command unit of rotation angles

The minimum command unit of the rotation angles (I, J, K, and R) of the tilted working plane indexing is 0.001 degree regardless of the increment system. By setting bit 2 (TRF) of parameter No. 11630 to 1, the minimum command unit of the rotation angles can be set to 0.00001 degree.

・System variables of skip positions during the tilted working plane indexing

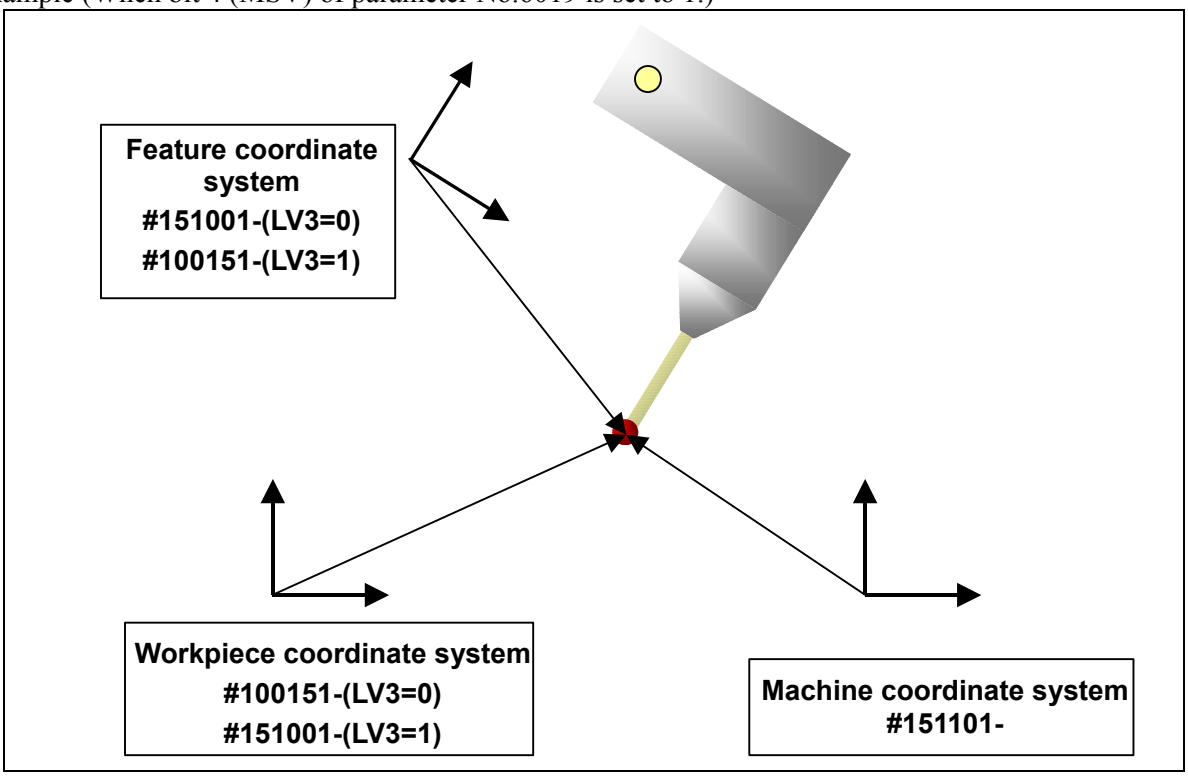
System variable number and coordinate system of skip are as Table22.6.1 (a). The coordinate system of #100105- and #151001- changes according to bit 5 (LV3) of parameter No.5400.

Table22.6.1 (a) Parameter LV3 and coordinate system of skip positions

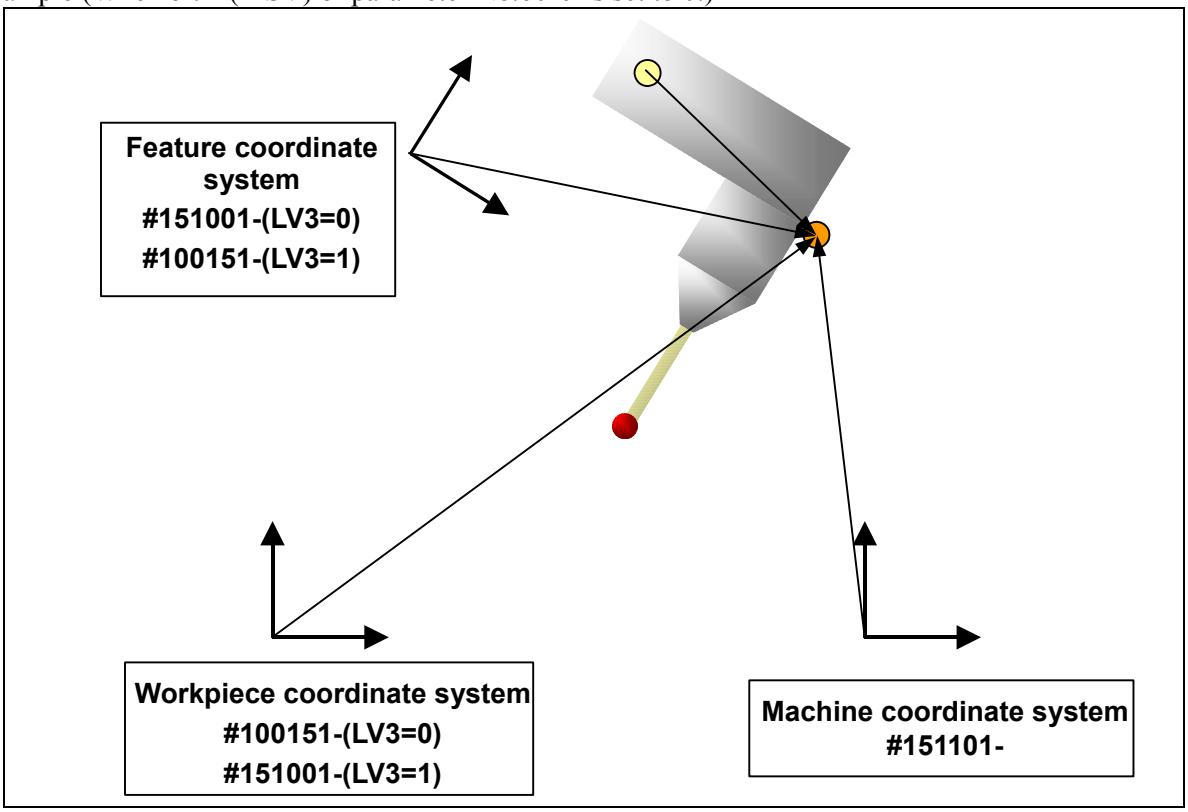
System variable number	Coordinate system of skip positions	
	Parameter LV3=0	Parameter LV3=1
#100151 -	Workpiece coordinate system	Feature coordinate system
#151001 -	Feature coordinate system	Workpiece coordinate system
#151150 -	Machine coordinate system	

Moreover, in case of machining center system, skip positions of tool tip position can be read by setting bit 4 (APZx) of parameter No. 6019.
When bit 4 (MSV) of parameter No.6019 is set to 0, system variable includes tool length compensation offset (control point position).
When bit 4 (MSV) of parameter No.6019 is set to 1, system variable does not include tool length compensation offset (tool tip position).

Example (When bit 4 (MSV) of parameter No.6019 is set to 1.)



Example (When bit 4 (MSV) of parameter No.6019 is set to 0.)



22.6.1.3 Tilted working plane indexing based on roll-pitch-yaw

Overview

With the tilted working plane indexing, coordinate system conversion by rotation about the X-axis, Y-axis, and Z-axis of a workpiece coordinate system in this order can be used (roll-pitch-yaw).
The order of rotary axes can be specified using address Q.

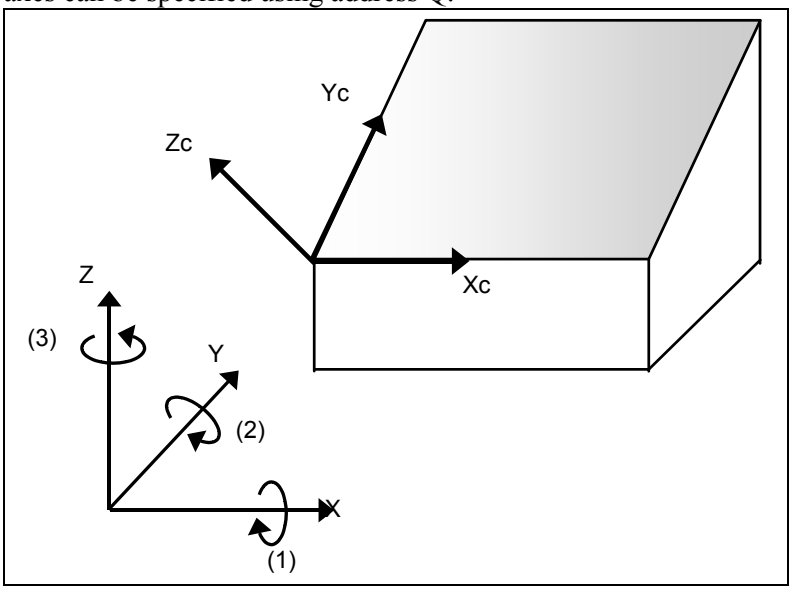


Fig. 22.6.1.3 (a)

Format

Format
G68.2 P1 Qq X_ Y_ Z_ Iα Jβ Kγ; Tilted working plane indexing **G69 ;** Cancel tilted working plane indexing (M series). **G69.1;** Cancel tilted working plane indexing (T series).
Explanation of symbols
Q : Order in which axes are rotated X_ Y_ Z_: Origin of a feature coordinate system I : Angle of rotation about the X-axis (roll angle) J : Angle of rotation about the Y-axis (pitch angle) K : Angle of rotation about the Z-axis (yaw angle)

The values of address Q and the order in which axes are rotated are shown below.

Table 22.6.1.3 (a)

	First rotation axis	Second rotation axis	Third rotation axis
Q 123	X axis	Y axis	Z axis
Q 132	X axis	Z axis	Y axis
Q 213	Y axis	X axis	Z axis
Q 231	Y axis	Z axis	X axis
Q 312	Z axis	X axis	Y axis
Q 321	Z axis	Y axis	X axis

⚠ CAUTION
1 When address Q is omitted, the X-axis, Y-axis, and Z-axis are rotated in this order. (equivalent to Q123)
2 When address Q is set to a value other than the above, alarm PS5457 is issued.

Explanation

Suppose that the coordinate system is rotated about (1) the X-axis, (2) the Y-axis, and (3) the Z-axis in this order.

A "workpiece coordinate system" rotated by angle α about the X-axis is "coordinate system 1".

"Coordinate system 1" rotated by angle β about the Y-axis is "coordinate system 2". "Coordinate system 2" rotated by angle γ about the Z-axis then shifted by (X_0,Y_0,Z_0) from the workpiece coordinate system origin is a "feature coordinate system".

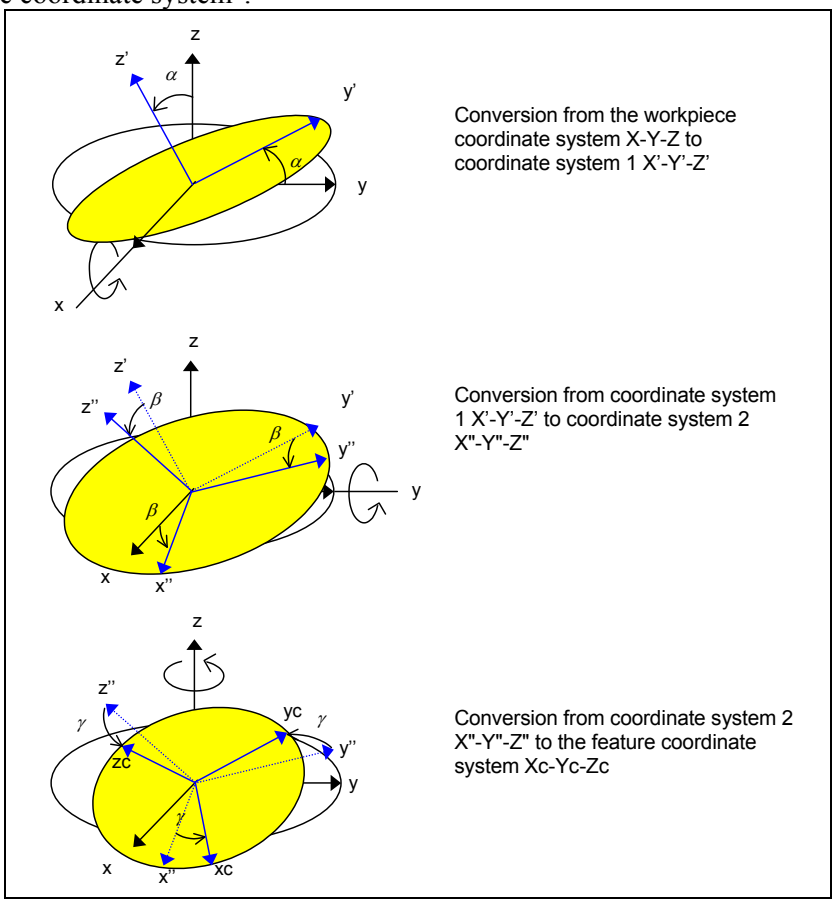


Fig. 22.6.1.3 (b)

Example

The example of a program when feature coordinate system like the figure below is used is shown below.

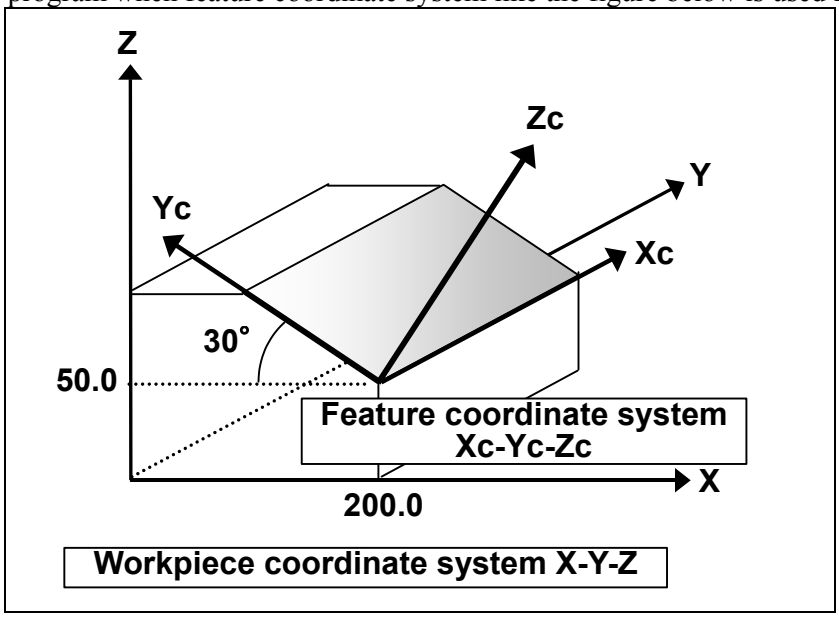


Fig. 22.6.1.3 (c)

- Feature coordinate system origin : (200.0, 0, 50.0)
- Order in which axes are rotated : X axis → Y axis → Z axis
- Angle of rotation about the X-axis : 30 degrees
- Angle of rotation about the Y-axis : 0 degree
- Angle of rotation about the Z-axis : 90 degrees

Example of a program

```
G68.2 P1 Q123 X200.0 Y0 Z50.0 I30.0 J0 K90.0 ;
G53.1 ;
  :
```

22.6.1.4 Tilted working plane indexing based on three points

Overview

With the tilted working plane indexing, a tilted working plane can be specified by specifying three points in a feature coordinate system.

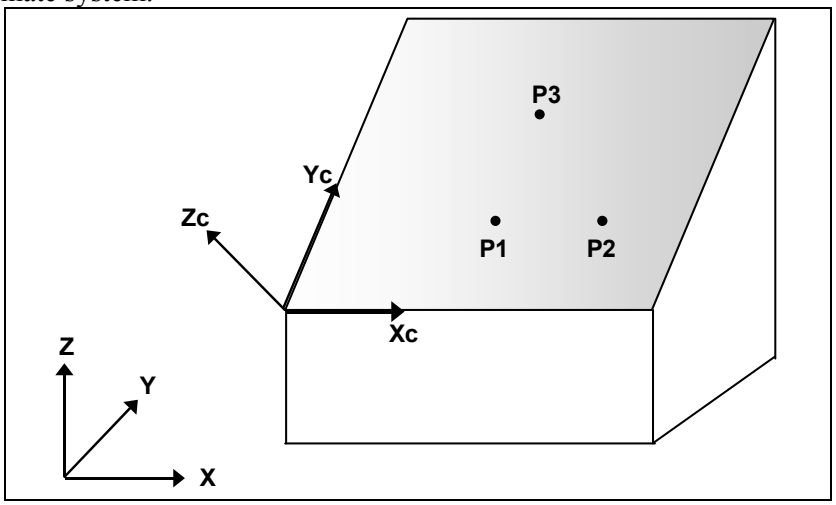


Fig. 22.6.1.4 (a)

Format

Format
G68.2 P2 Q0 X x_0 Y y_0 Z z_0 Rα ; **G68.2 P2 Q1 X x_1 Y y_1 Z z_1 ;** **G68.2 P2 Q2 X x_2 Y y_2 Z z_2 ;** **G68.2 P2 Q3 X x_3 Y y_3 Z z_3 ; Tilted working plane indexing** **G69 ; Cancel tilted working plane indexing (M series).** **G69.1; Cancel tilted working plane indexing (T series).**
Explanation of symbols
Q0 X x_0 Y y_0 Z z_0 : Amount of shift from the first point to the origin of a feature coordinate system By default, this value is 0. Q1 X x_1 Y y_1 Z z_1 : First point. (origin of a feature coordinate system) Q2 X x_2 Y y_2 Z z_2 : Second point. Q3 X x_3 Y y_3 Z z_3 : Third point. R : Angle of rotation about the Z-axis of a feature coordinate system By default, this value is 0. Any block in the G68.2 P2 command can be specified.

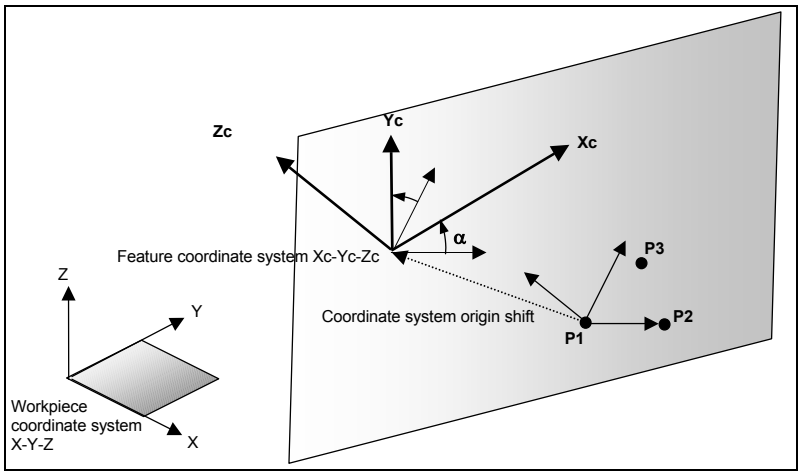


Fig. 22.6.1.4 (b)

⚠ CAUTION

1 Three G68.2P2 commands (Q1, Q2, and Q3) determine a tilted plane. If the G68.2P2 commands are interrupted, alarm PS5457 is issued.
2 When any of the following conditions is met or address Q is set to a value other than the above, alarm PS5457 is issued.
(1) When two or more points are duplicate (the plane is not determined.)
(2) When three points are placed on a line (the plane is not determined.)
(3) When the distance between a line passing two of the three points and the remaining point is smaller than the setting of parameter No. 11220 (the plane is unstable.)

Explanation

- Determination of a feature coordinate system

Three entered points are named P1, P2, and P3 in the order of entry.
The P1-to-P2 direction is defined as the X-axis of a feature coordinate system. Among the directions that are on the plane containing the three points and are normal to the X-axis of the feature coordinate system, the direction that makes a smaller angle with the P1→P3 vector is defined as the Y-axis of the feature coordinate system. The Z-axis of the feature coordinate system is defined according to the right-handed system.

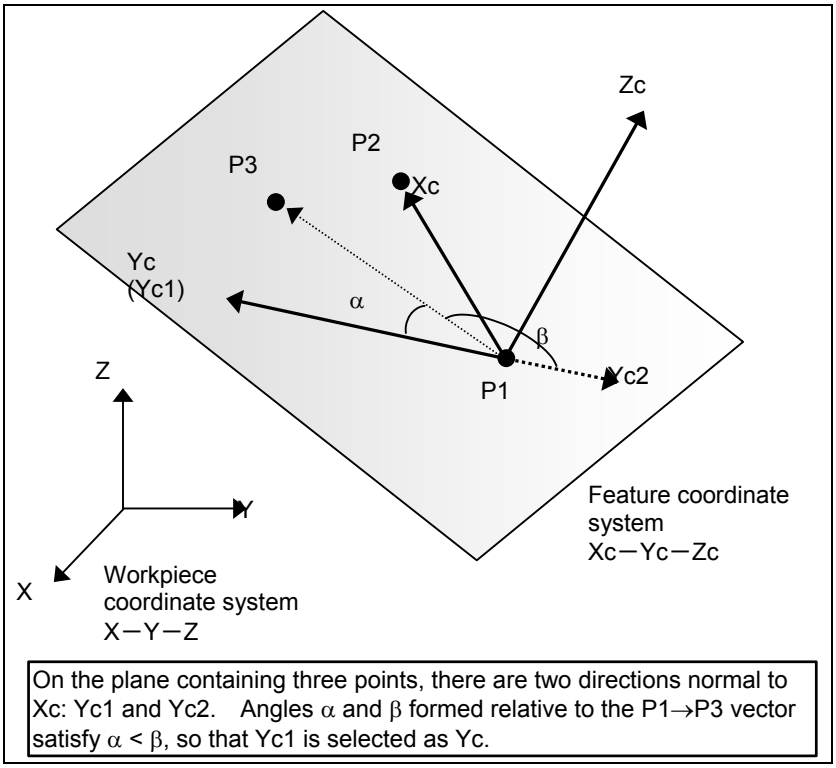


Fig. 22.6.1.4 (c)

- Origin of the feature coordinate system

The origin of the feature coordinate system is the first specified point P1.
By setting an origin shift amount (G68.2 P2 Q0 X_Y_Z_), the origin of the feature coordinate system is shifted by (X,Y,Z) from P1. Specify (X,Y,Z) in the feature coordinate system.

- Angular displacement R

The angular displacement R is positive for clockwise rotation viewed in the Z-axis direction in the feature coordinate system.

Example

The example of a program when feature coordinate system like the figure below is used is shown below.

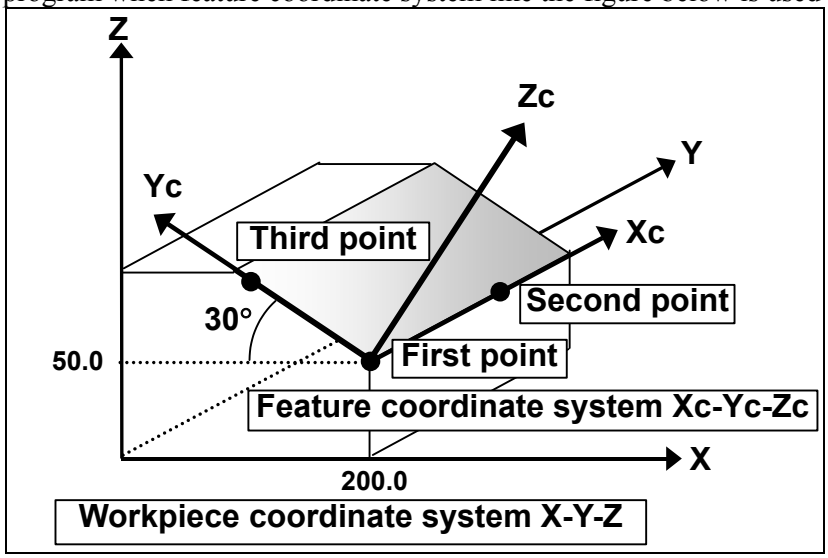


Fig. 22.6.1.4 (d)

- First point (origin of a feature coordinate system) : (200.0, 0, 50.0)
- Second point : (200.0, 100.0, 50.0)
- Third point : (26.795, 0, 150.0)

Example of a program

```
G68.2 P2 Q1 X200.0 Y0 Z50.0 ;
G68.2 P2 Q2 X200.0 Y100.0 Z50.0 ;
G68.2 P2 Q3 X26.795 Y0 Z150.0 ;
G53.1 ;
 . . .
```

22.6.1.5 Tilted working plane indexing based on two vectors

Overview

With the tilted working plane indexing, a tilted working plane can be specified by specifying an X-axis direction vector and a Z-axis direction vector in the feature coordinate system.

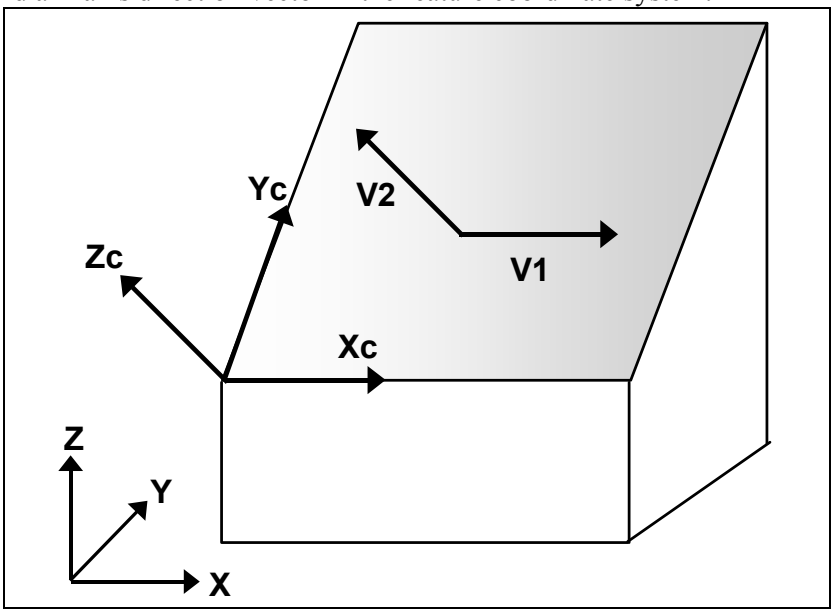


Fig. 22.6.1.5 (a)

Format

Format
G68.2 P3 Q1 X_ Y_ Z_ Iα_1 Jβ_1 Kγ_1 ; **G68.2 P3 Q2 Iα_2 Jβ_2 Kγ_2 ; Tilted working plane indexing** **G69 ; Cancel tilted working plane indexing (M series).** **G69.1; Cancel tilted working plane indexing (T series).**
Explanation of symbols
X_ Y_ Z_: Origin of a feature coordinate system (specifiable in the Q1 block) Q1 Iα_1 Jβ_1 Kγ_1 : Feature coordinate system X-axis direction relative to the workpiece coordinate system (first vector) Q2 Iα_2 Jβ_2 Kγ_2 : Feature coordinate system Z-axis direction relative to the workpiece coordinate system (second vector)

⚠ CAUTION
1 The two G68.2P3 commands (Q1 and Q2) determine a tilted plane. The G68.2P3 commands are interrupted, alarm PS5457 is issued.
2 If the angle between the two vectors is 5 degrees or more off the 90 degrees, alarm PS5457 is issued.
3 If (I, J, K) is set to a 0 vector, alarm PS5457 is issued.

Explanation

- Determination of a feature coordinate system

The first vector is defined as the X-axis of the feature coordinate system, and the second vector is defined as the Z-axis of the feature coordinate system. The Y-axis of the feature coordinate system is defined according to the right-handed system.

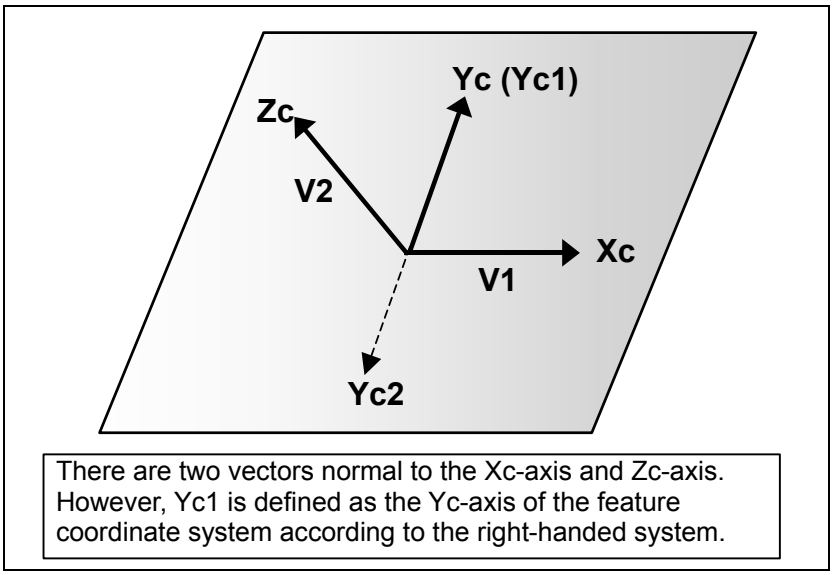


Fig. 22.6.1.5 (b)

- When the first and second vectors are not normal

When the first vector and second vector are not normal, the orthographical vector from the second vector to plane P normal to the first vector is defined as the Z-axis of the feature coordinate system.

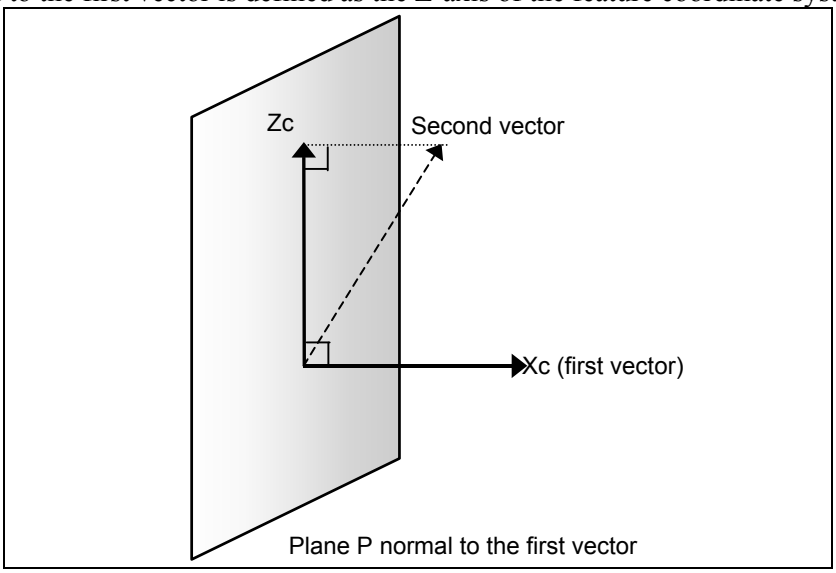


Fig. 22.6.1.5 (c)

Example

The example of a program when feature coordinate system like the figure below is used is shown below.

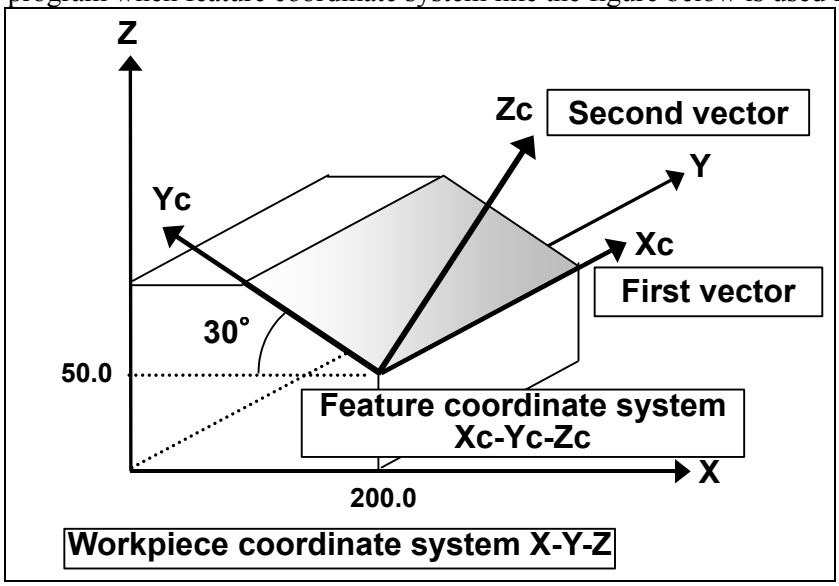


Fig. 22.6.1.5 (d)

Origin of a feature coordinate system : (200.0, 0, 50.0)
X-axis direction of a feature coordinate system (First vector) : (0, 1.0, 0)
Z-axis direction of a feature coordinate system (Second vector) : (100.0, 0, 173.205)

Example of a program

```
G68.2 P3 Q1 X200.0 Y0 Z50.0 I0 J1.0 K0 ;
G68.2 P3 Q2 I100.0 J0 K173.205 ;
G53.1 ;
 . . .
```

22.6.1.6 Tilted working plane indexing based on projection angles

Overview

With the tilted working plane indexing, a tilted working plane can be specified based on projection angles.

A plane determined by vector A and vector B produced by rotating the X-axis vector and Y-axis vector of the workpiece coordinate system is defined to be a tilted working plane.

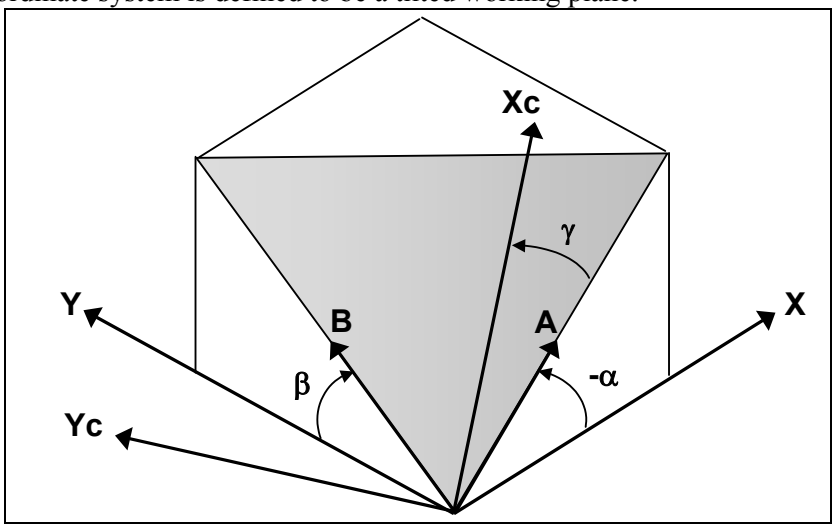


Fig. 22.6.1.6 (a)

Format

Format
G68.2 P4 X_ Y_ Z_ Iα Jβ Kγ; Tilted working plane indexing **G69 ; Cancel tilted working plane indexing (M series).** **G69.1; Cancel tilted working plane indexing (T series).**
Explanation of symbols
X_ Y_ Z_ : Origin of a feature coordinate system α : Angle by which the X-axis vector is rotated about the Y-axis of the workpiece coordinate system β : Angle by which the Y-axis vector is rotated about the X-axis of the workpiece coordinate system γ : Angle of rotation about the Z-axis of the feature coordinate system

Explanation

- Determination of a feature coordinate system

The X-axis direction vector of the workpiece coordinate system rotated by α about the Y-axis of the workpiece coordinate system is defined as vector A. The Y-axis direction vector of the workpiece coordinate system rotated by β about the X-axis of the workpiece coordinate system is defined to be vector B.

The direction normal to plane P containing vector A and vector B (direction of the outer product of A × B) is defined to be the Z-axis direction of the feature coordinate system.

Vector A rotated by γ about the Z-axis of the feature coordinate system is defined to be the X-axis direction of the feature coordinate system. The Y-axis of the feature coordinate system is defined according to the right-handed system.

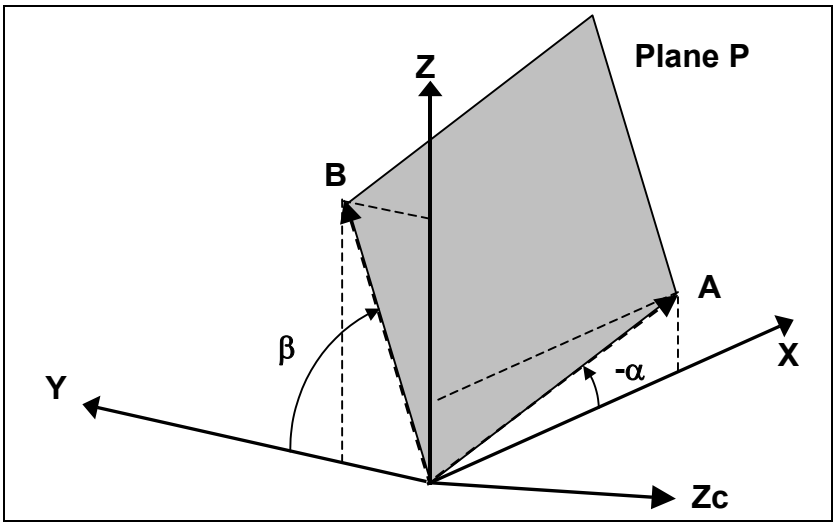


Fig. 22.6.1.6 (b)

By the third command angle α and second command angle β, the Z-axis of the feature coordinate system are determined.

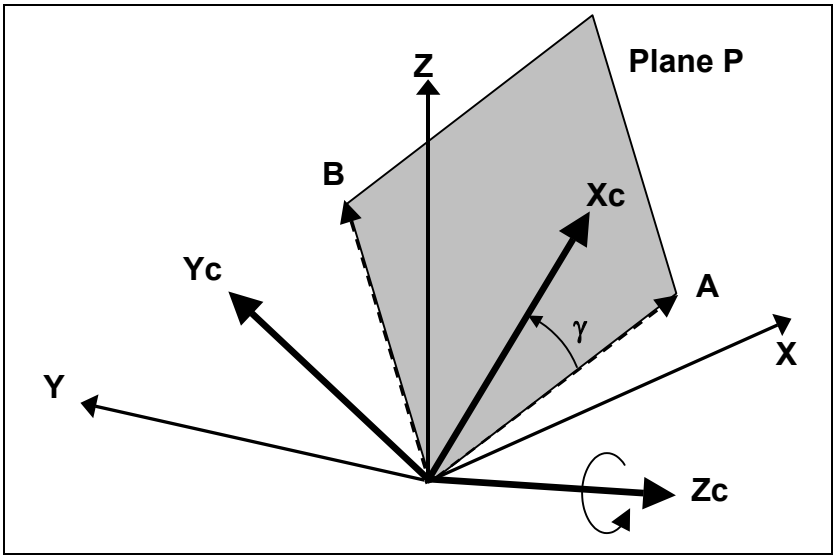


Fig. 22.6.1.6 (c)

By the third command angle γ, the X-axis and Y-axis of the feature coordinate system are determined.

> ⚠ **CAUTION**
> When vector A and vector B are considered to be parallel with each other (when the angle formed by the two vectors is smaller than 1°), alarm PS5457 is issued.

Example

The example of a program when feature coordinate system like the figure below is used is shown below.

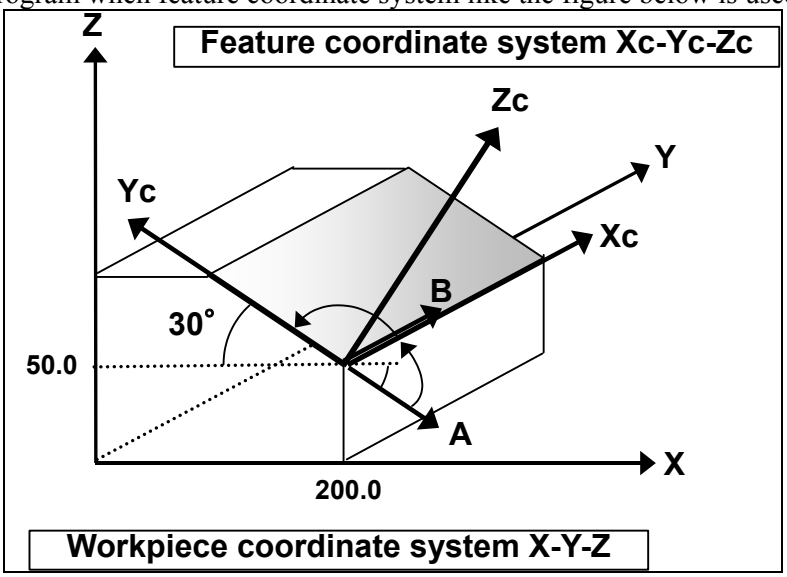


Fig. 22.6.1.6 (d)

- Origin of a feature coordinate system : (200.0, 0, 50.0)
- Angle by which the X-axis vector is rotated about the Y-axis of the workpiece coordinate system : 30 degrees
- Angle by which the Y-axis vector is rotated about the X-axis of the workpiece coordinate system : 0 degree
- Angle of rotation about the Z-axis of the feature coordinate system : 90 degrees

Example of a program

```
G68.2 P4 X200.0 Y0 Z50.0 I30.0 J0 K90.0 ;
G53.1 ;
      :
```

22.6.1.7 Tilted working plane indexing by tool axis direction

Overview

By specifying G68.3, a coordinate system (feature coordinate system) where the tool axis direction is the +Z-axis direction can be automatically specified. When a feature coordinate system is used, a program for cutting a hole or pocket in a plane tilted relative to the workpiece coordinate system can be made simpler. This function can automatically generate a feature coordinate system that is normal to the tool direction.

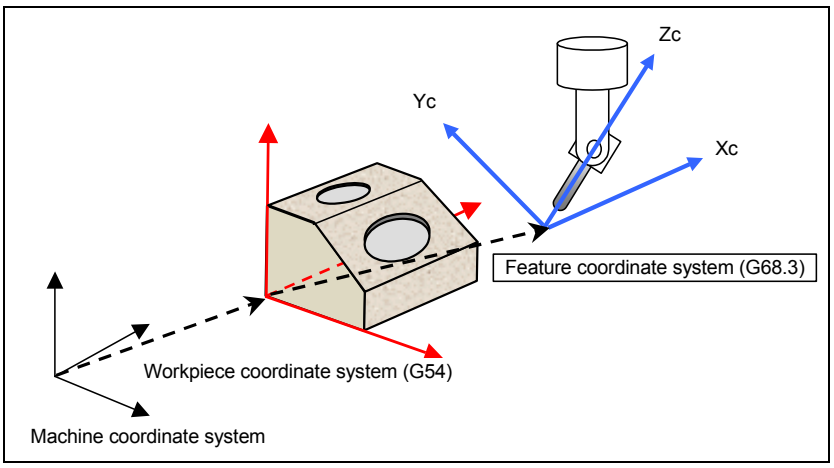


Fig. 22.6.2 (a) Feature coordinate system

When G68.3 is specified in a block, the coordinate system for programming is changed to a feature coordinate system. All commands after the block are regarded as commands in the feature coordinate system until G69 is specified.

Format

Format
G68.3 X x_0 Y y_0 Z z_0 Rα ; Tilted working plane indexing
G68.3 P1 X x_0 Y y_0 Z z_0 ; Tilted working plane indexing
G69 ; Cancel tilted working plane indexing (M series).
G69.1; Cancel tilted working plane indexing (T series).
Explanation of symbols
X,Y,Z : Origin of a feature coordinate system (absolute) By default, the current position is used as the origin of the feature coordinate system.
R : Angular displacement about the Z-axis in the feature coordinate system. The default is 0°.
P1 : The feature coordinate system is defined corresponding to tool rotation axis position.

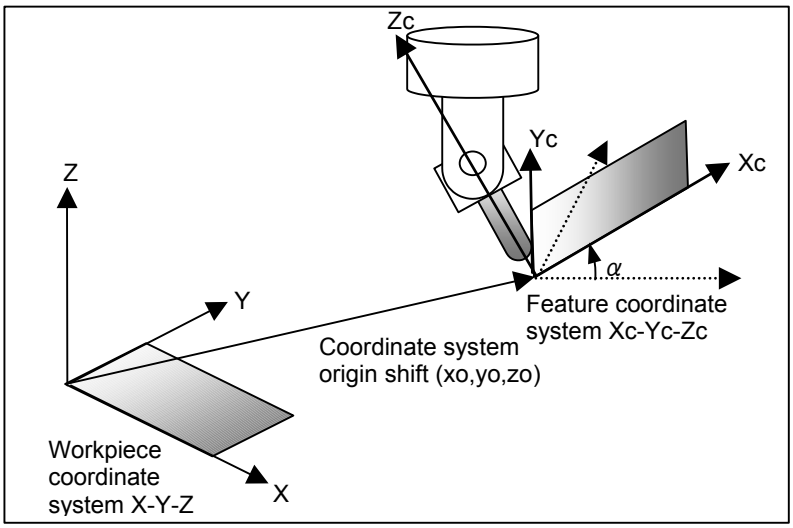


Fig. 22.6.2 (b) G68.3 command

Explanation

- Feature coordinate system

By specifying G68.3, a feature coordinate system with the tool axis direction being the +Z-axis direction can be created. The tool axis direction means the tool axis direction based on the rotation axis position reached by automatic operation or manual operation.

- Origin of a feature coordinate system

The origin of a feature coordinate system must be specified using an absolute command.
Even in the incremental command mode, the specified origin is regarded as an absolute position.
When 1 address or 2 addresses are omitted in X, Y, Z, alarm PS5457 is issued.
When the origin of a feature coordinate system is not specified, the position when G68.3 is specified is used as the origin.

- Determination of a feature coordinate system

Determination of a feature coordinate system depends on whether P1 command is present.

(1) When G68.3 block does not include P1 command (G68.3)

When G68.3 is specified, the tool axis direction vector ($\vec{T}$) represents the +Z direction ($\vec{Zc}$) of the feature coordinate system.

The vector normal to a plane formed by the +Z direction ($\vec{Zc}$) of the feature coordinate system and the vertical axis direction vector ($\vec{P}$) (parameter No. 12321) represents the +X direction ($\vec{Xc}$) of the feature coordinate system.

Expression: $\vec{Xc} = \vec{P} \times \vec{Zc}$

The vector normal to the +Z direction ($\vec{Zc}$) of the feature coordinate system and the +X direction ($\vec{Xc}$) of the feature coordinate system represents the +Y direction ($\vec{Yc}$) of the feature coordinate system.

Expression: $\vec{Yc} = \vec{Zc} \times \vec{Xc}$

When R is commanded, a coordinate system rotated by R around Zc from the above-mentioned coordinate system is the feature coordinate system.

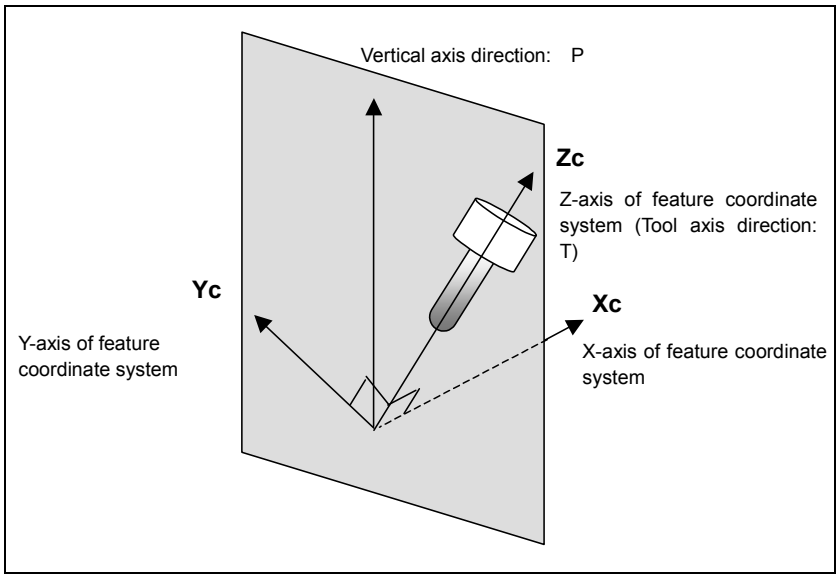


Fig. 22.6.2 (c) Determination of a feature coordinate system

When the tool axis direction vector ($\vec{T}$) is parallel with the vertical axis direction vector ($\vec{P}$) (parameter No. 12321) (when the angle between the vector ($\vec{T}$) and the vector ($\vec{P}$) is equal to or less than the value of parameter No. 12322), the feature coordinate system Xc-Yc-Zc is as indicated below. By specifying angular displacement R, a feature coordinate system rotated about the Z-axis of this coordinate system can be specified.

Table 22.6.2 (a)

Parameter No. 12321	Z-axis of feature coordinate system Zc	X-axis of feature coordinate system Xc	Y-axis of feature coordinate system Yc
1	+X direction	+Y direction	+Z direction
2	+Y direction	+Z direction	+X direction
3	+Z direction	+X direction	+Y direction

When 0 is set in parameter No. 12321, the vertical axis direction is the reference tool axis direction (parameter No. 19697).
If a value other than 0 through 3 is set in parameter No. 12321, the PS5459 alarm is issued.

> ⚠ **CAUTION**
> Tool axis direction is Z-axis direction of feature coordinate system regardless of the reference tool axis direction (parameter No. 19697).

- Angular displacement R

Angular displacement R is positive when a rotation is made clockwise viewed in the Z-axis direction of the feature coordinate system. The range of angular displacement R is: $0.0° \leq R \leq 360.0°$.

(2) When G68.3 block includes P1 command (G68.3 P1)

"G68.3 P1" command defines the feature coordinate system corresponding to tool rotation axis position. The direction of the feature coordinate system is the direction of reference coordinate system rotated by tool rotation axes. The direction of this feature coordinate system is the same as the direction of tool axis direction feed / tool axis right-angle direction feed of 3-dimensional manual feed when the parameter FLL (No.12320#1) is set to zero.

The reference coordinate system of feature coordinate system (the feature coordinate system that is defined when absolute coordinate system of tool rotation axes is zero) is as follows by the parameter (No.19697) for reference tool axis direction.
The feature coordinate system defined by "G68.3 P1" command is the coordinate system that the reference coordinate system is rotated by tool rotation axis position and parameter RA (No.19698), RB (No.19699).

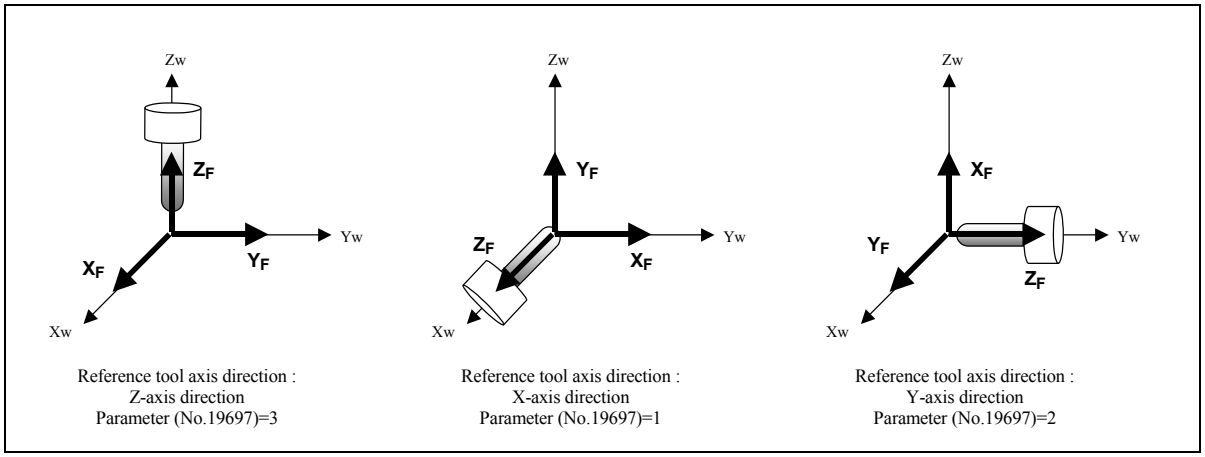
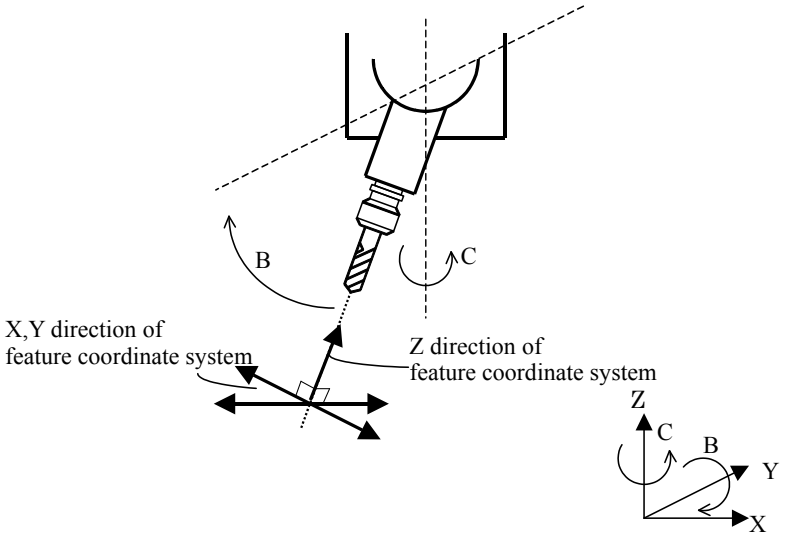


Fig. 22.6.2 (d) The reference coordinate system of feature coordinate system (G68.3 P1)



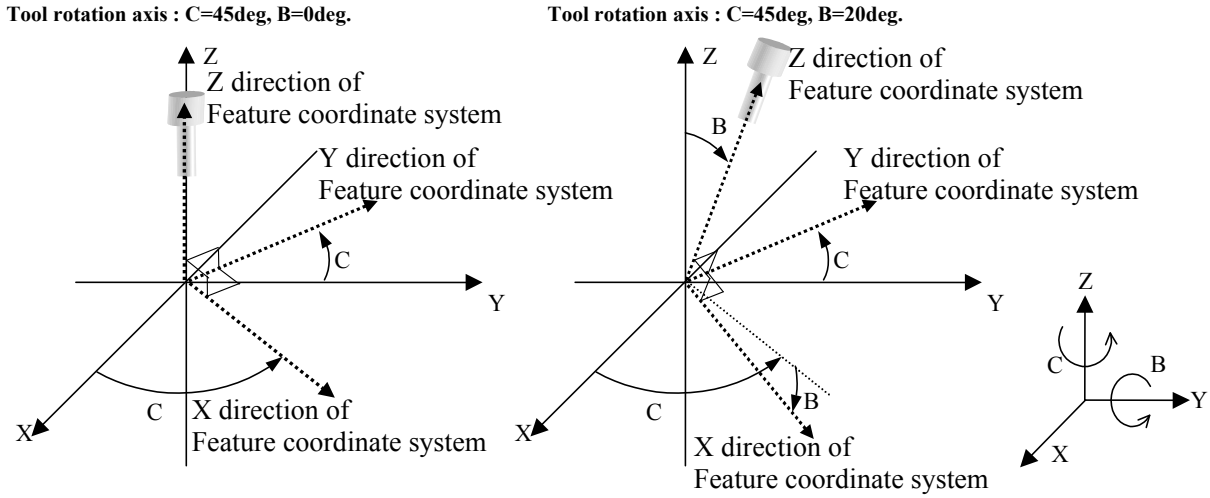


Fig. 22.6.2 (e) Example that reference tool axis direction is Z direction

- Machine of table rotation type

On a machine of table rotation type, the tool direction remains unchanged. So, a feature coordinate system based on the reference tool axis direction (parameter No. 19697) is set. However, the origin specification of the feature coordinate system and angular displacement R about the Z-axis are valid.

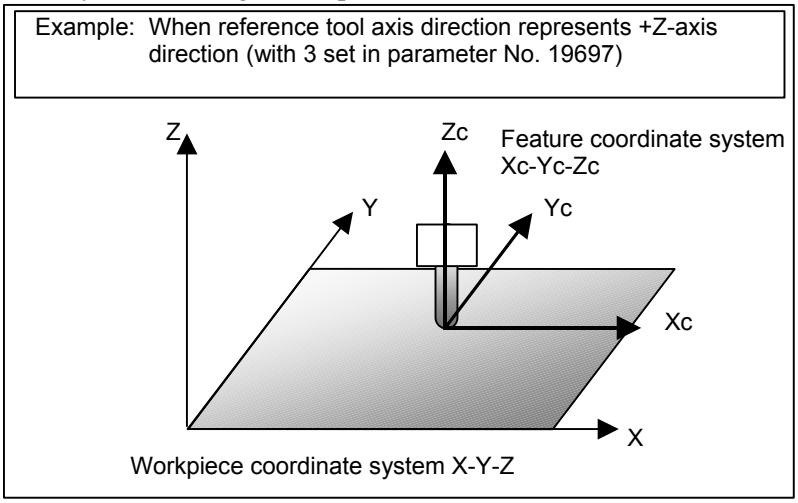


Fig. 22.6.2 (f)

- Use in combination with tool length compensation

The G68.3 command can be specified even during tool length compensation.

NOTE
Do not specify G68.3 during workpiece setting error compensation. Otherwise, alarm PS0437 is issued.

- Example of operation

An example of operation on a machine of tool rotation type is given below.
The machine configuration is "BC type reference tool axis Z-axis".

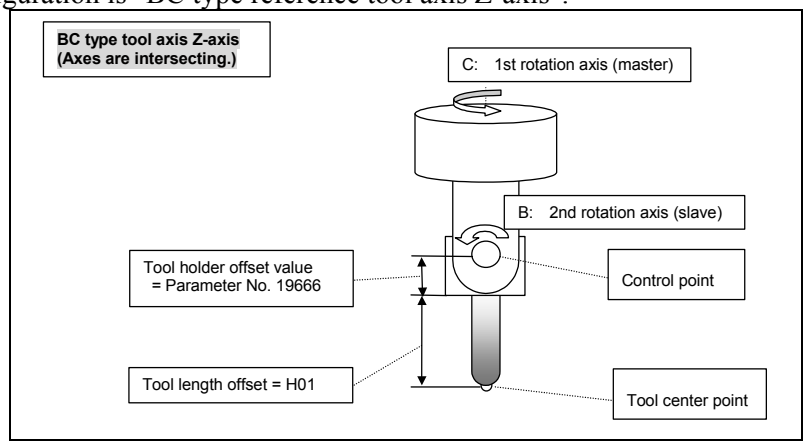


Fig. 22.6.2 (g)

Sample program 1

```
O0100 ;
N1 G55 ;
N2 G90 G01 X0Y0Z50.0 F1000 ;
N3 G43 H01 X0 Y0 Z0 ;
N4 B-45.0 ;
N5 G68.3 ;
:
N6 G69 ;
:
```

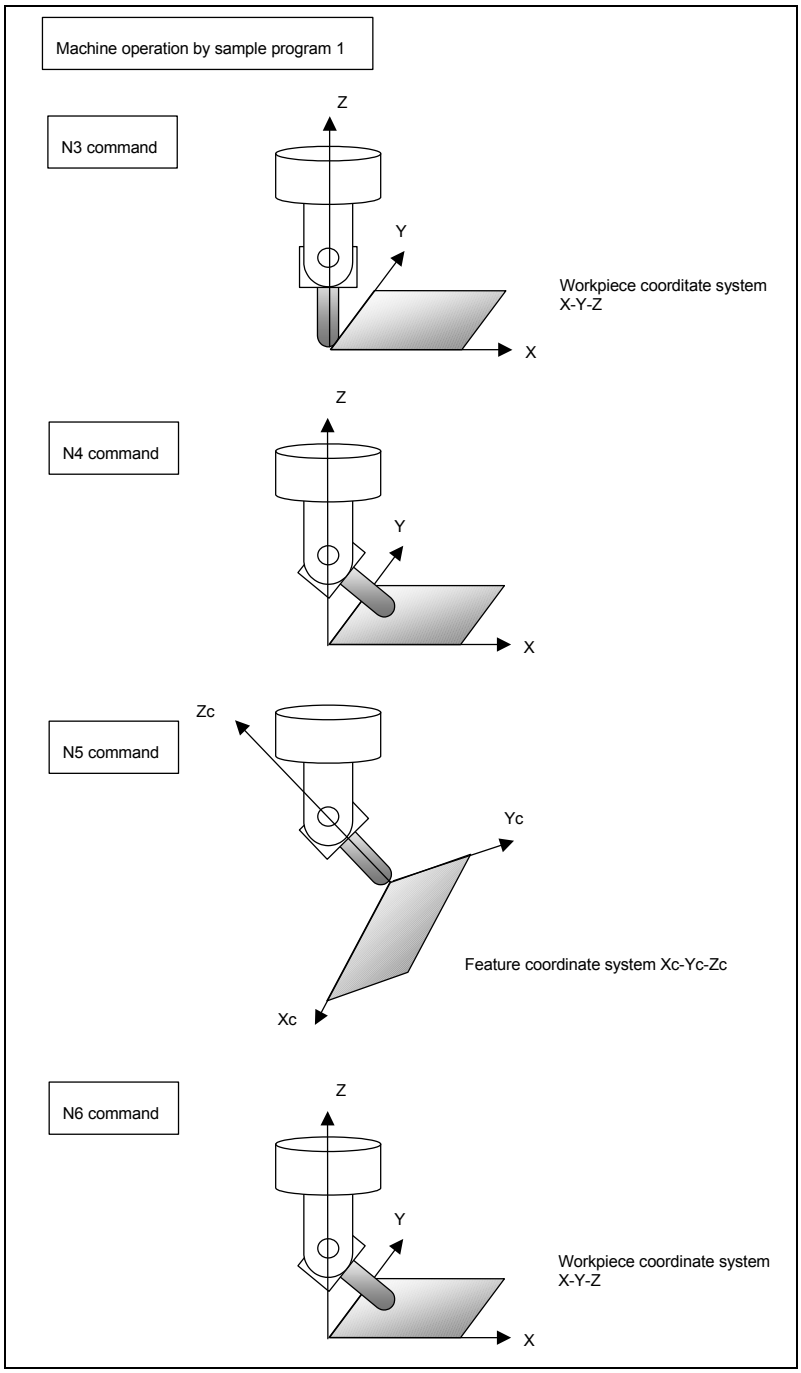
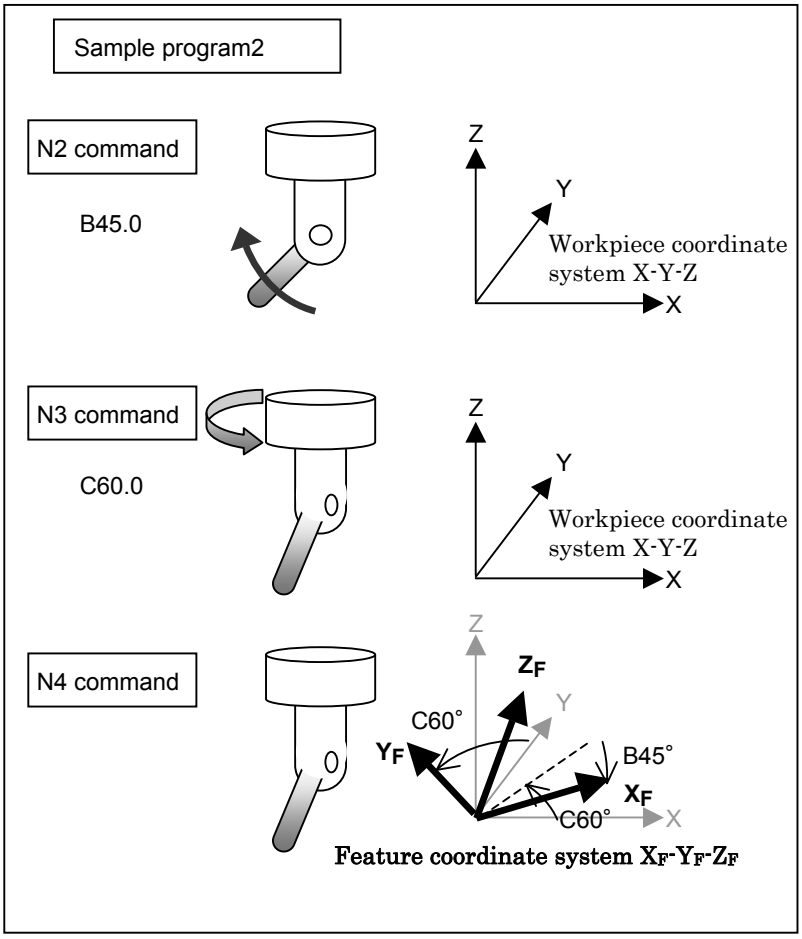


Fig. 22.6.2 (h)

N3 block: Performs tool length compensation in the workpiece coordinate system.
The tool center point moves to the origin of the workpiece coordinate system.
N4 block: Tilts the tool.
N5 block: Sets a feature coordinate system where the tool axis direction is the Z-axis direction and the tool center point is placed at the origin.
N6 block: Cancels the feature coordinate system to return to the workpiece coordinate system.

Sample program 2

```
O0100 ;
N1 G54 G90 G00 B0 C0 ;
N2 B45.0 ;
N3 C60.0 ;
N4 G68.3 P1 X0 Y0 Z0 ;
```



N2 block: Tilts the tool. (B45 deg)
N3 block: Tilts the tool. (C60 deg)
N4 block: The direction of the reference coordinate system of feature coordinate system is the direction of workpiece coordinate system because the reference tool axis direction is Z direction. The feature coordinate system is the coordinate system that the reference coordinate system is rotated by 45 deg. around Y direction and 60 deg. around Z direction.

X_F : The direction that X direction of workpiece coordinate system is rotated by 60 deg. around Z direction after by 45 deg. around Y direction of workpiece coordinate system.

Y_F : The direction that Y direction of workpiece coordinate system is rotated by 60 deg. round Z direction of workpiece coordinate system.

Z_F : The direction that Z direction of workpiece coordinate system is rotated by 60 deg. around Z direction after by 45 deg. around Y direction of workpiece coordinate system.

- Multiple G68.3

After the tool axis direction is changed in G68.3 mode, by specifying G68.3, a new feature coordinate system where the tool axis direction is the +Z-axis direction can be specified.

Example of operation

An example of operation on a machine of tool rotation type is given below.
The machine configuration is "AC type reference tool axis Z-axis".

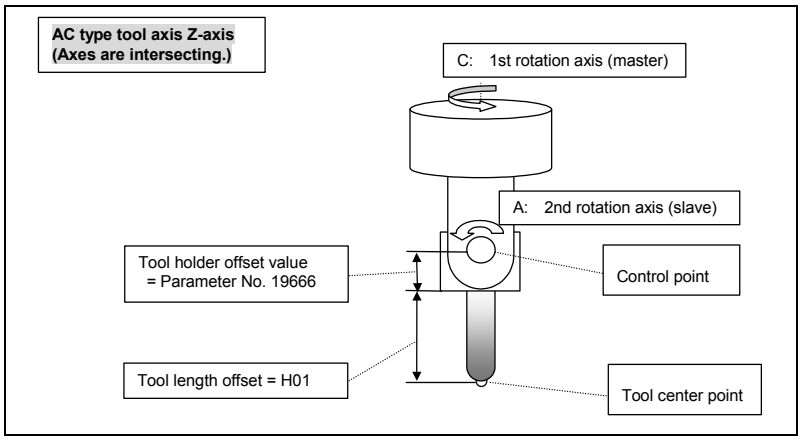


Fig. 22.6.2 (i)

Sample program 2

```
O0200 ;
N1 G55 ;
N2 G01 A90.0 F1000 ;
N3 G68.3 X0 Y0 Z0 R0;
:
N4 X10.0 Y0 Z0 ;
N5 C90.0;
N6 G68.3 X10.0 Y0 Z0 ;
:
N7 G69 ;
:
```

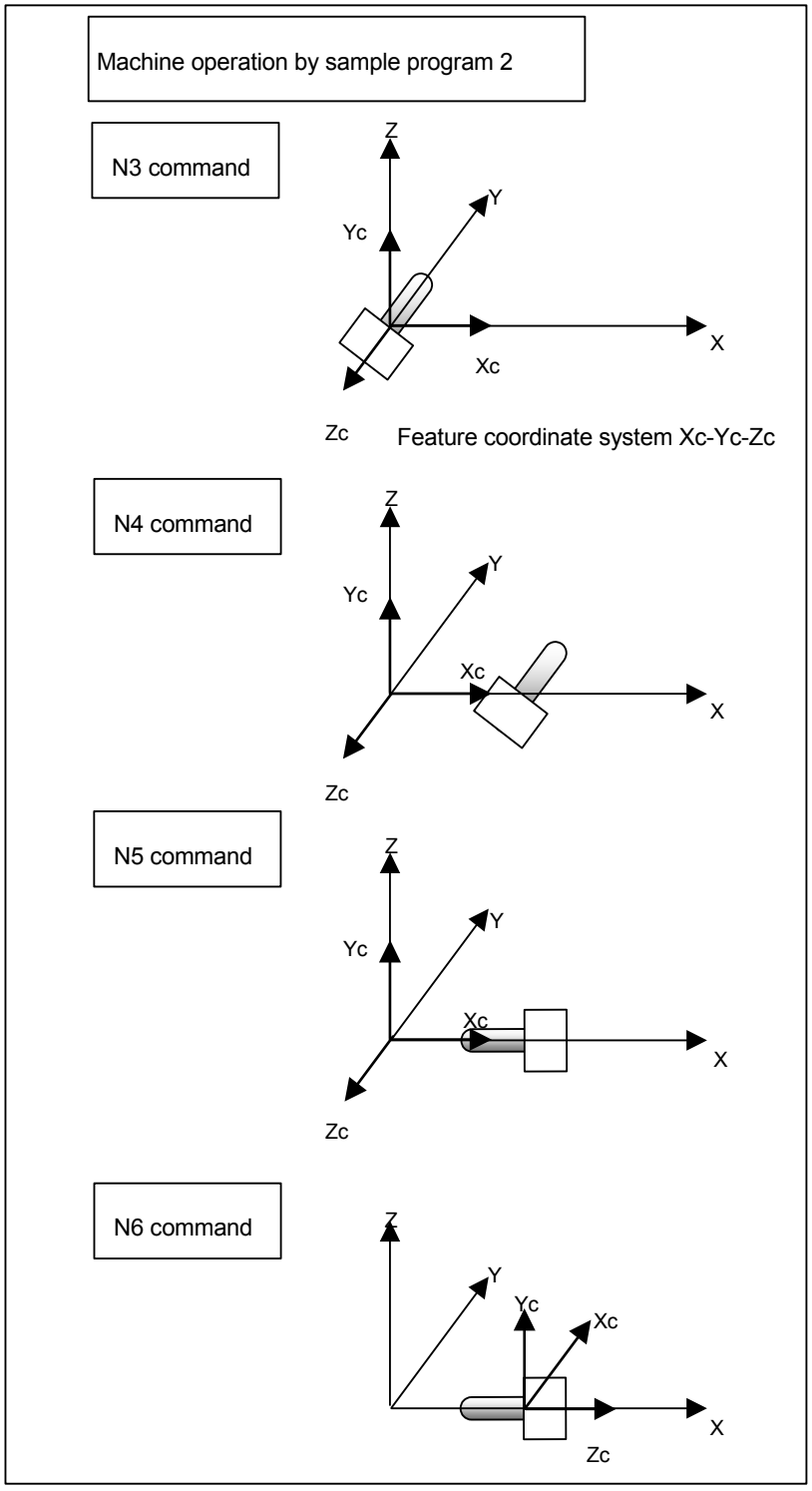


Fig. 22.6.2 (j)

N3 block: Sets a feature coordinate system according to the tool direction.
N4 block: Specifies coordinates in the feature coordinate system.
N5 block: Changes the tool direction.
N6 block: Sets a feature coordinate system according to the tool direction.

22.6.2 Multiple command of tilted working plane indexing

22.6.2.1 Absolute multiple command

By additionally specifying G68.2 in the tilted working plane indexing mode, a feature coordinate system produced by additionally applying coordinate system conversion to the workpiece coordinate system can be set. The workpiece coordinate system is resumed by specifying G69.
This function is enabled by setting bit 0 (MTW) of parameter No. 11221.

Format

The format of the tilted working plane indexing (G68.2) is applicable.
Specify the origin of a feature coordinate system in the workpiece coordinate system.

NOTE
Before specifying G68.2, cancel tool length compensation and tool radius compensation. If G68.2 is specified during tool length compensation or tool radius compensation, alarm PS5462 is issued.

Example of operation

An example of operation on a tool rotation type machine is explained below.
The machine configuration is "BC type with the reference tool axis being the Z-axis".

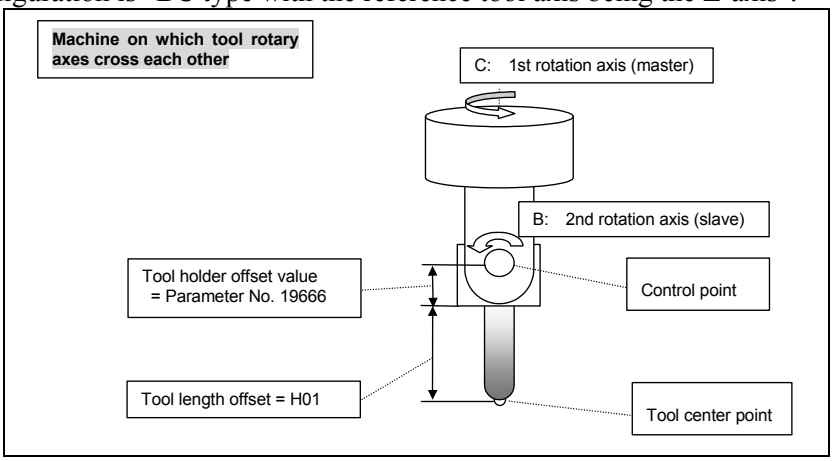


Fig. 22.6.2.1 (a)

Sample program 2

```
O0100 ;
N1 G55 ;
N2 G90 G01 X20.0 Y5.0 Z0 F1000 ;
N3 G68.2 X20.0 Y5.0 Z0 I0 J90.0 K0 ;
N4 G53.1 ;
  :
N5 X-15.0 Y0 Z-15.0 ;
N6 G68.2 X5.0 Y20.0 Z0 I90.0 J90.0 K0 ;
N7 G53.1;
  :
N8 G69 ;
  :
```

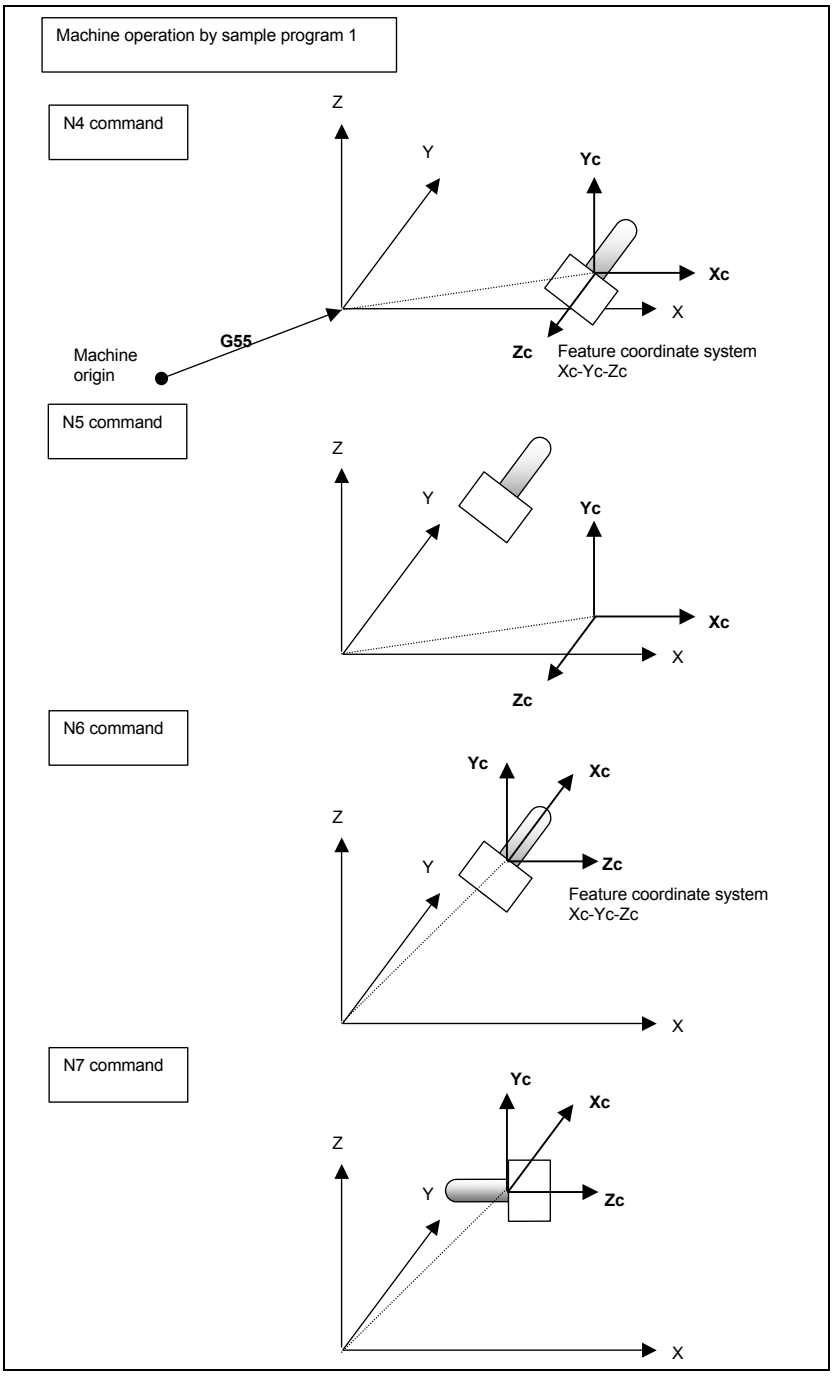


Fig. 22.6.2.1 (b)

N4 block: Rotates the tool in the Z-axis direction in the feature coordinate system.
N5 block: Specifies coordinates in the feature coordinate system.
N6 block: Sets a new feature coordinate system.
N7 block: Rotates the tool in the Z-axis direction in the new feature coordinate system.

22.6.2.2 Incremental multiple command

By specifying G68.4, coordinate system conversion can be applied to the currently set feature coordinate system.
This function is enabled by setting bit 0 (MTW) of parameter No. 11221.

Format

The format of the tilted working plane indexing (G68.2) is applicable.
Specify the origin of a feature coordinate system in the immediately preceding feature coordinate system.

Table 22.6.2.2 (a)

Specification method	Incremental multiple command
Eulerian angle	G68.4
Roll-pitch-yaw	G68.4 P1
Three points	G68.4 P2
Two vectors	G68.4 P3
Projection angles	G68.4 P4

NOTE
1 Before specifying G68.4, cancel tool length compensation and tool radius compensation. If G68.4 is specified during tool length compensation or tool radius compensation, alarm PS5462 is issued.
2 Do not specify G68.4 during workpiece setting error compensation. Otherwise, alarm PS0437 is issued.

Example of operation

An example of operation on a tool rotation type machine is explained below.
Rotary axis C rotates about the Z-axis (master axis).
Rotary axis B rotates about the Y-axis (slave axis).

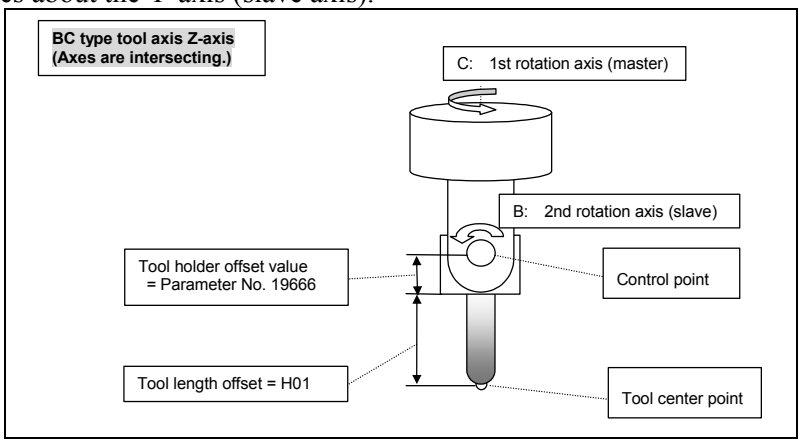


Fig. 22.6.2.2 (a)

Sample program 3

```
O0200 ;
N1 G55 ;
N2 G90 G01 X20.0 Y5.0 Z0 F1000 ;
N3 G68.2 X20.0 Y5.0 Z0 I0 J90.0 K0 ;
N4 G53.1 ;
  :
N5 X-15.0 Y0 Z-15.0 ;
N6 G68.4 X-15.0 Y0 Z-15.0 I90.0 J90.0 K-90.0 ;
```

```
N7 G53.1;
  :
N8 G69 ;
  :
```

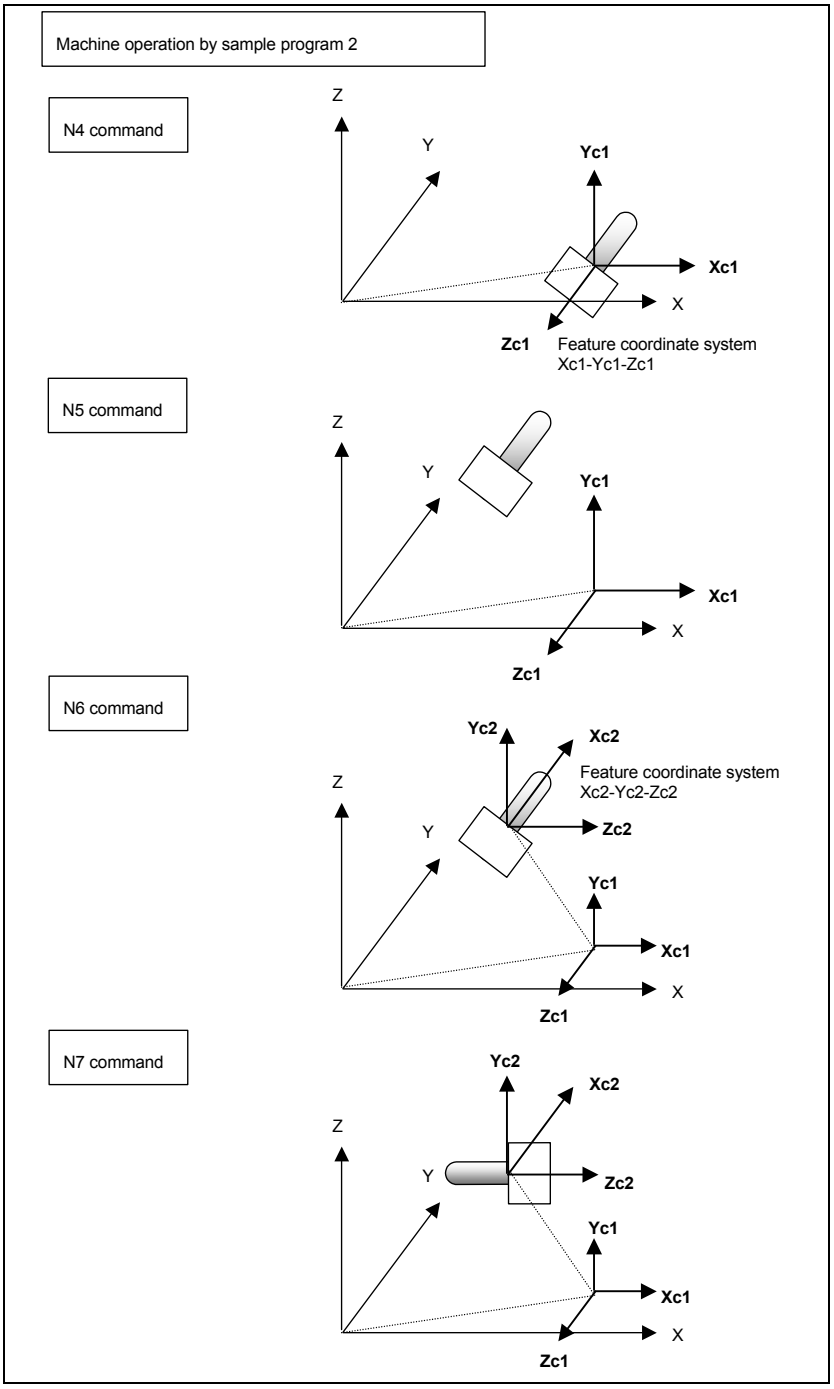


Fig. 22.6.2.2 (b)

N4 block: Rotates the tool in the Z-axis direction in the feature coordinate system.
N5 block: Specifies coordinates in the feature coordinate system.
N6 block: Applies coordinate system conversion to the feature coordinate system based on Eulerian angle to set a new feature coordinate system.
N7 block: Rotates the tool in the Z-axis direction in the new feature coordinate system.

22.6.3 Tool Axis Direction Control

22.6.3.1 Tool axis direction control

G53.1 automatically specifies the +Z direction of the feature coordinate system as the tool axis direction.

Example of operation

The following gives an operation example in the machine configuration below.

- Table rotation type
- Master axis: About the Y-axis (B axis)
- Slave axis: About the Z-axis (C axis)
- Reference tool axis direction: Z direction
- Increment system for the rotation axis: 1/10(IS-C)

Program example 1

G68.2 I90.0 J0.0005 K-90.0 (rotation by 0.0005 degree about the Y-axis)
G53.1

The rotation axis position after the G53.1 command is as follows:

- Bit 2 (TFR) of parameter No. 11630 is set to 0 (minimum command unit of the rotation angles: 0.001 degree):
  B axis: 0.0010 degree
  C axis: 0.0000 degree
- Bit 2 (TFR) of parameter No. 11630 is set to 1 (minimum command unit of the rotation angles: 0.00001 degree):
  B axis: 0.0005 degree
  C axis: 0.0000 degree

Program example 2

G68.2 I90.0 J10 K-90.0 (rotation by (minimum command unit of the rotation angles × 10) about the Y-axis)
G53.1

The rotation axis position after the G53.1 command is as follows:

- Bit 2 (TFR) of parameter No. 11630 is set to 0 (minimum command unit of the rotation angles: 0.001 degree):
  B axis: 0.0100 degree
  C axis: 0.0000 degree
- Bit 2 (TFR) of parameter No. 11630 is set to 1 (minimum command unit of the rotation angles: 0.00001 degree):
  B axis: 0.0001 degree
  C axis: 0.0000 degree

- Tool rotation type machine

The following paragraphs describe several cases of the tool rotation type machine operation.

Operation description 1:

When G43 (tool length compensation) is specified for a machine with its axes crossing one another

The G53.1 command, when specified after the G68.2 command, automatically controls the rotary axis in such a way that the tool axis will be oriented in the +Z direction of the feature coordinate system.

Example)

```
O100 (Sample Program1) ;
N1 G55 ;
N2 G90 G01 X0 Y0 Z30.0 F1000 ;
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0 ;
N4 G01 X0 Y0 Z30.0 F1000 ;
N5 G53.1 ;
N6 G43 H01 X0 Y0 Z0 ;
N7 . . .
```

In this example, the "BC type tool axis Z-axis" is used as the machine configuration.
In addition, the tool axis, tool rotation axis B, and tool rotation axis C cross one another.

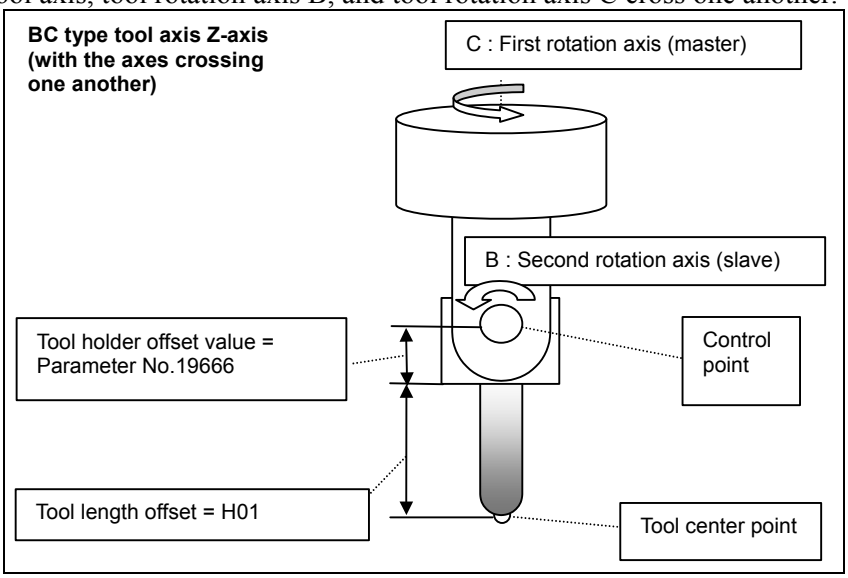


Fig. 22.6.3 (c)

Block N3 : Defines a feature coordinate system in the workpiece coordinate system.
Block N4 : Shifts the control point to point Z30.0 in the feature coordinate system.
Block N5 : Exerts automatic control over the rotary axes.
Block N6 : Performs tool length compensation in the feature coordinate system.
The tool center point is shifted to the origin of the feature coordinate system.

Fig. 22.6.3 (d) shows the behavior of the machine when it runs sample program 1.

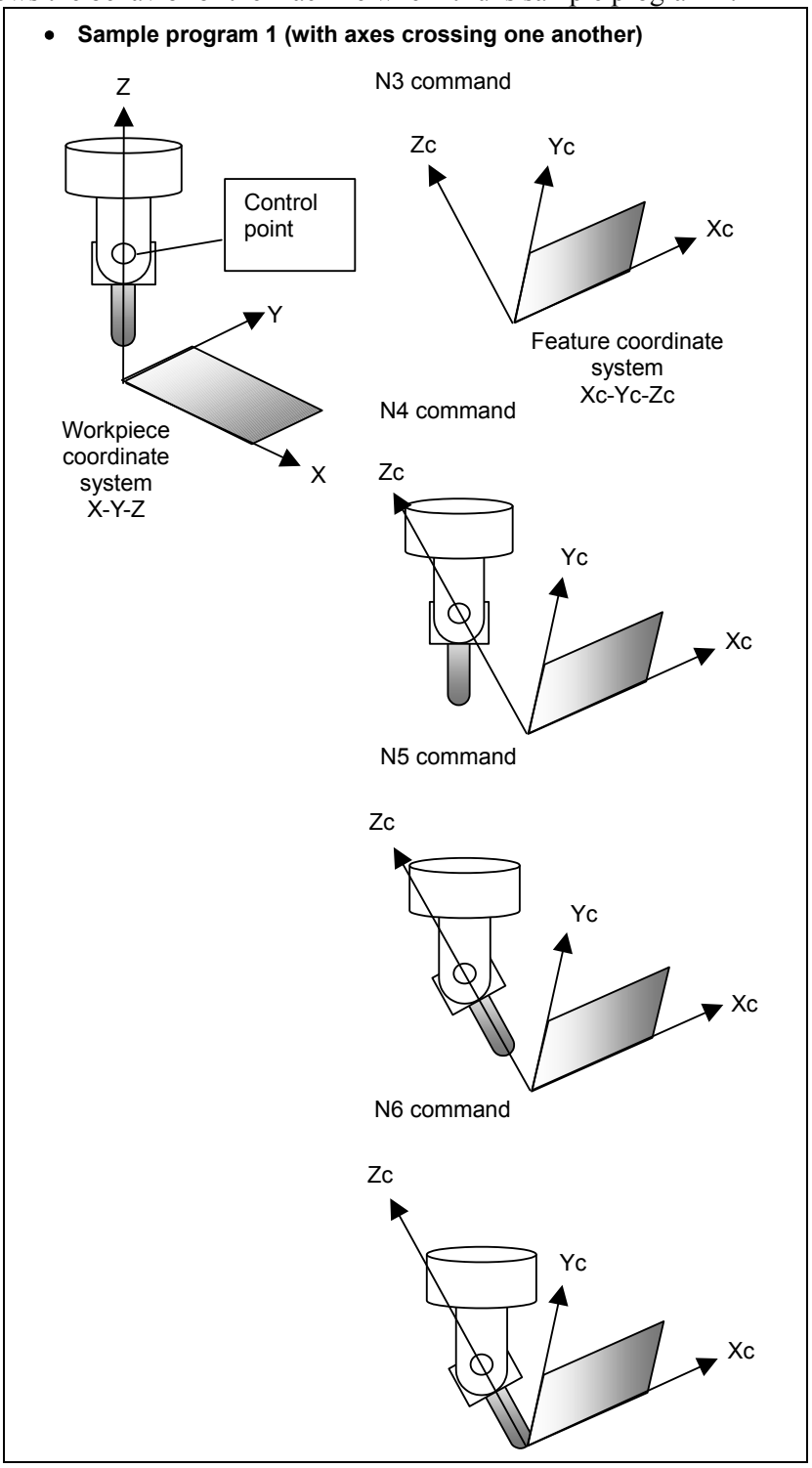


Fig. 22.6.3 (d) Tool axis direction control 1

Operation description 2:

When G43 (tool length compensation) is specified for a machine with no axis crossing

Here is the case where no axis of the machine crosses any other axis.
It is assumed that sample program 1 is used.

In this example, the "BC type tool axis Z-axis" is used as the machine configuration.
It is assumed, however, that the tool axis does not cross the B-axis while the B-axis and C-axis cross each other.

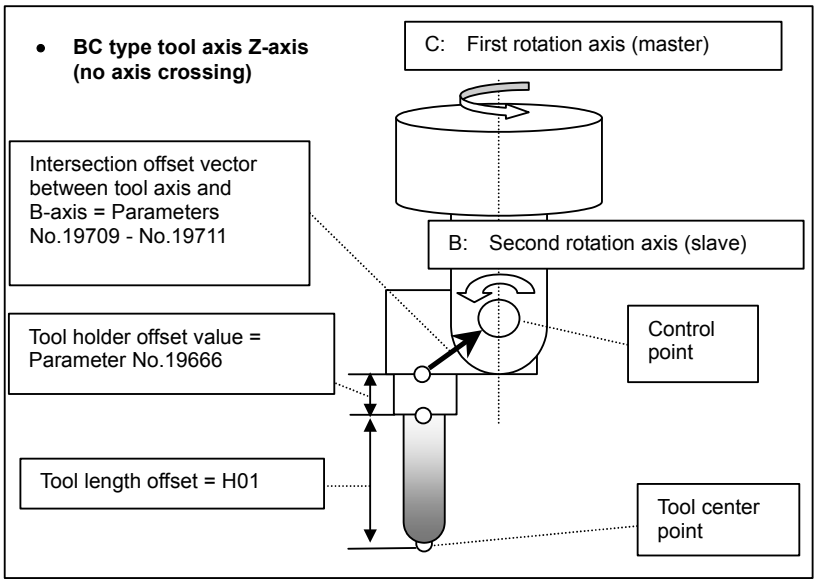


Fig. 22.6.3 (e)

Block N4 : Shifts the control point to point Z30.0 in the feature coordinate system.
Block N5 : Exerts automatic control over the rotary axes.
Block N6 : An intersection offset vector between the tool axis and the B-axis with automatic control for rotary axes taken into consideration is output in the feature coordinate system.
Performs tool length compensation in the feature coordinate system.
The tool center point is shifted to the origin of the feature coordinate system.

This is also true when the B-axis does not cross the C-axis.
For explanations about the offset to be applied when the B-axis does not cross the C-axis, see the descriptions about parameters No.19712, No.19713, and No.19714.

Fig. 22.6.3 (f) shows the behavior of the machine when it runs sample program 1.

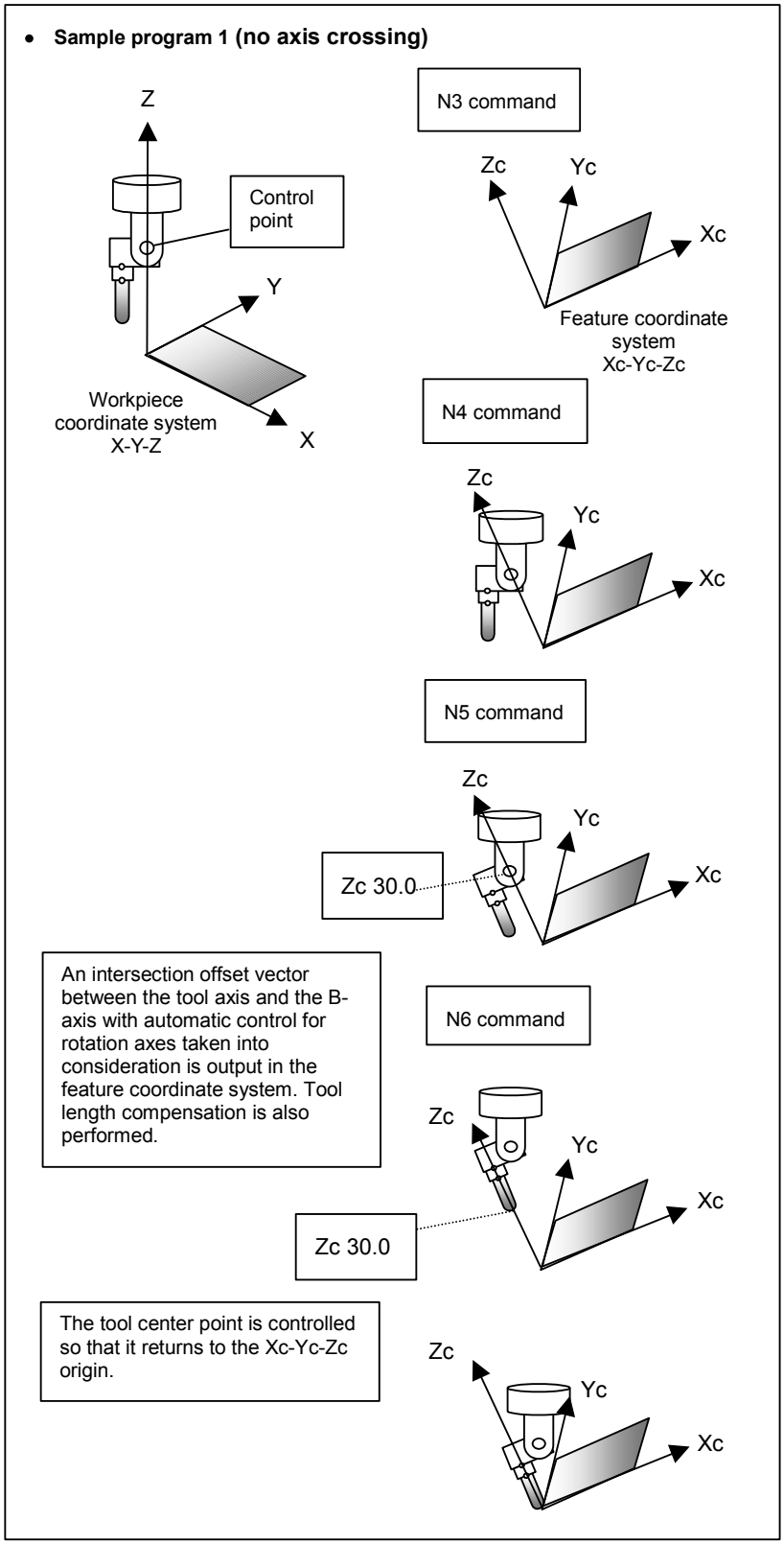


Fig. 22.6.3 (f) Tool axis direction control 2

Operation description 3:
When no G43 (tool length compensation) command is specified or if no G53.1 (tool axis direction control) command is specified

Sample program 2 of O200 is equivalent to sample program 1 except that sample program 2 has no tool length compensation command (G43).
Example)

```
O200 (Sample Program2) ;
N1 G55 ;
N2 G90 G01 X0 Y0 Z30.0 F1000 ;
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0 ;
N4 G01 X0 Y0 Z0 F1000 ;
N5 G53.1 ;
N6 . . . ;
```

In this example, the "BC type tool axis Z-axis" is used as the machine configuration.
The case in which the axes cross one another and the case in which no axis crosses any other axis are described.
Fig. 22.6.3 (g) shows the behavior of the machine when it runs sample program 2.

Sample program 3 of O300 is equivalent to sample program 1 except that sample program 3 has no tool axis direction control command (G53.1).
Example)

```
O300 (Sample Program3) ;
N1 G55 ;
N2 G90 G01 X0 Y0 Z30.0 F1000 ;
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0 ;
N4 G01 X0 Y0 Z0 F1000 ;
N5 G43 H01 ;
N6 . . . ;
```

In this example, the "BC type tool axis Z-axis" is used as the machine configuration.
The case in which the axes cross one another and the case in which no axis crosses any other axis are described.
Tool length compensation is applied in the +Z-axis direction of the feature coordinate system.
Fig. 22.6.3 (h) shows the behavior of the machine when it runs sample program 3.

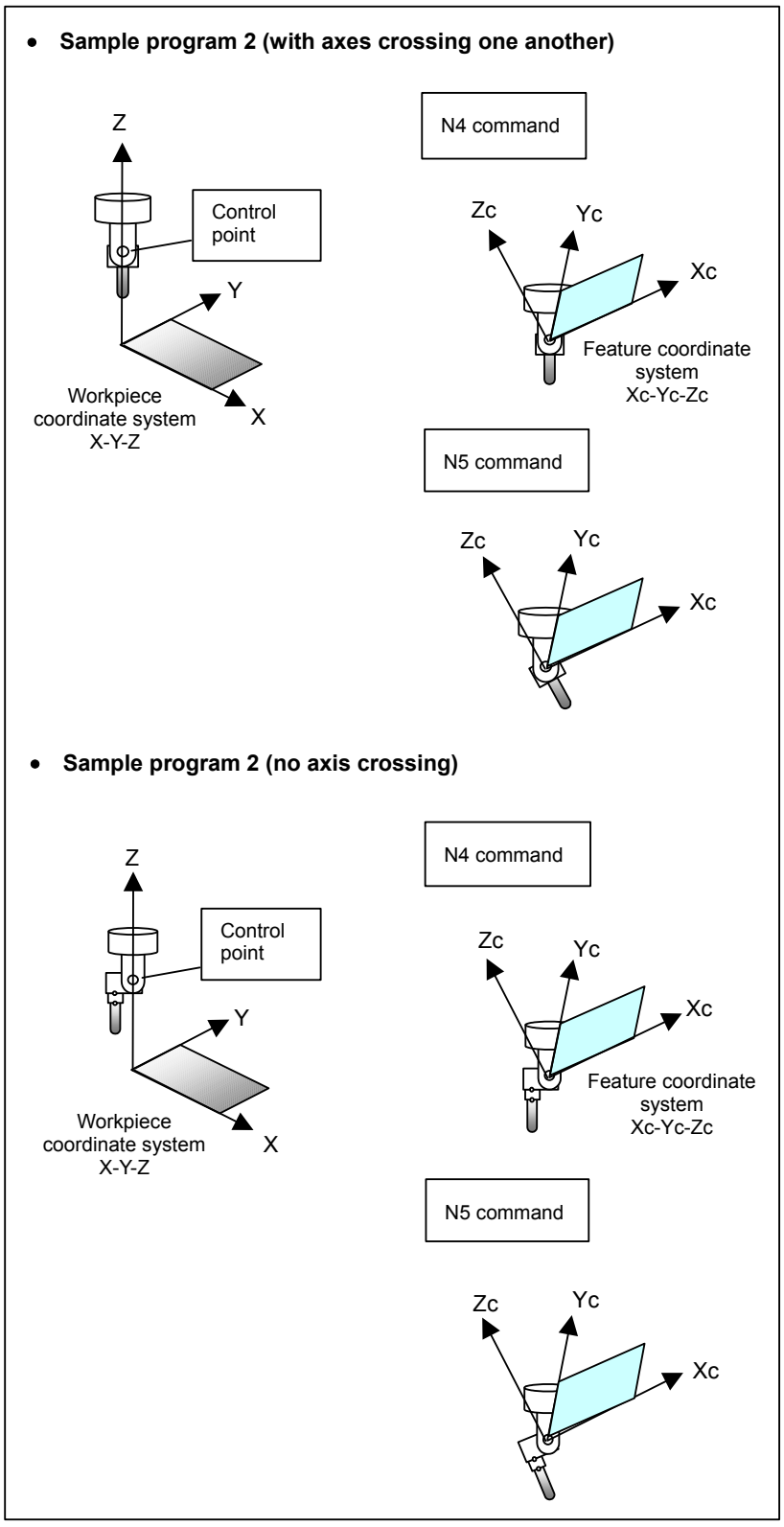


Fig. 22.6.3 (g) When the tool length compensation command is not specified

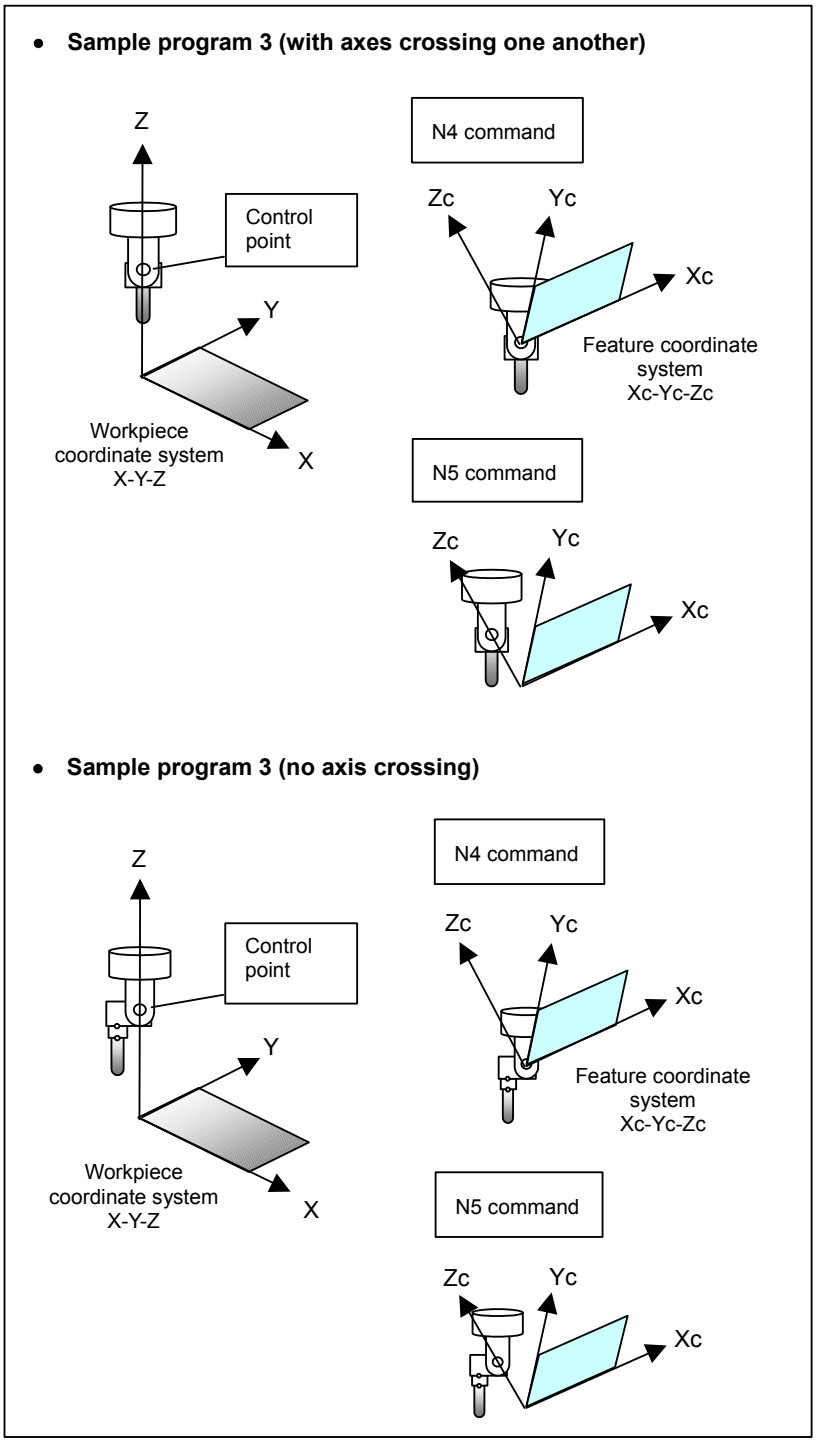


Fig. 22.6.3 (h) When the tool axis direction control command is not specified

- Composite type machine
Basic operation

This function is also available for a composite type machine in which the tool head rotates on the tool rotation axis and the table rotates on the table rotation axis.
The feature coordinate system Xc-Yc-Zc is set in the workpiece coordinate system based on the coordinate system origin shift (xo, yo, zo) and the Euler's angle.
Given the A-axis and B-axis shown in Fig. 22.6.3 (i), control is performed in such a way that the A-axis rotates until Zc comes in the X-Z plane and the B-axis is controlled so that the tool axis is oriented toward the +Z-axis direction of the feature coordinate system.

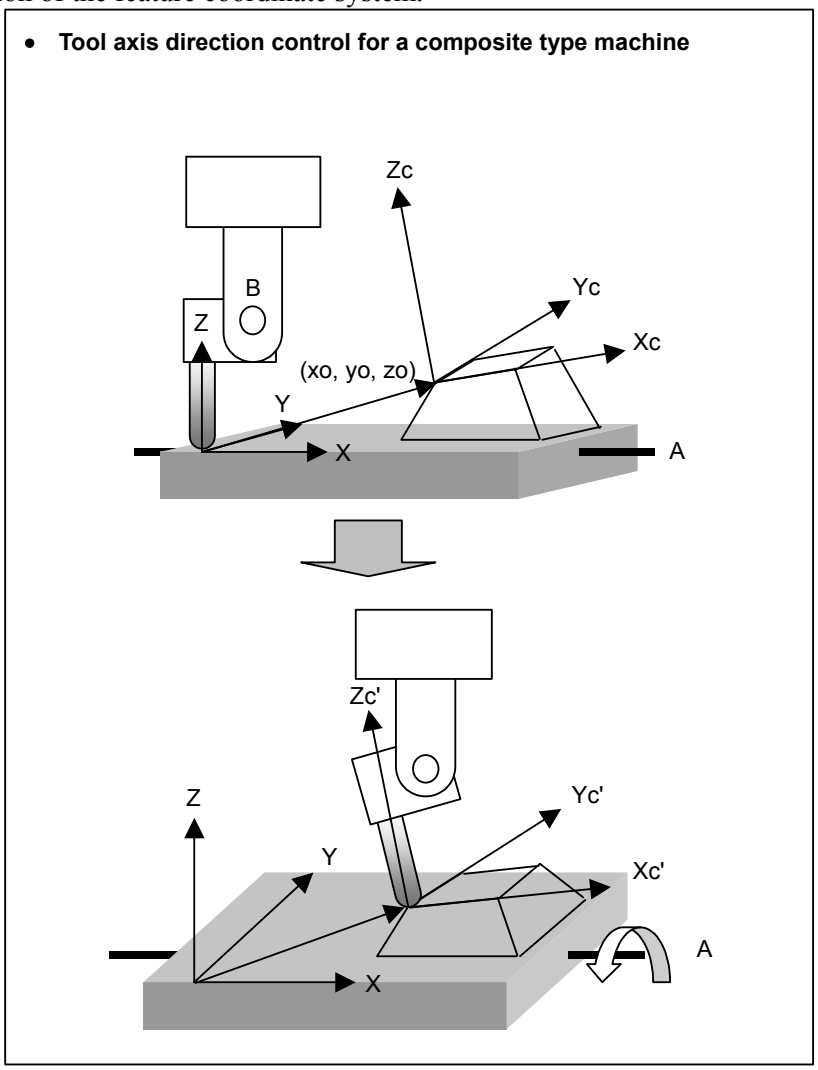


Fig. 22.6.3 (i) Composite type machine

- Feature coordinate system with the table rotated by G53.1 (tool axis direction control)

The composite type machine shown in Fig. 22.6.3 (i) is explained as an example.
If the table rotates by the tool axis direction control command (G53.1), the feature coordinate system (called the first feature coordinate system), which is set in the workpiece coordinate system by the tilted working plane indexing (G68.2), rotates as much as the table rotates.
The feature coordinate system that has rotated is called the second feature coordinate system.
Once G53.1 is specified, the subsequent machining commands are assumed to be specified in the second feature coordinate system. (See Fig. 22.6.3 (j).)
In the composite type machine, the specified feature coordinate system (the first feature coordinate system) may differ from the feature coordinate system to be used for machining (the second feature coordinate system).

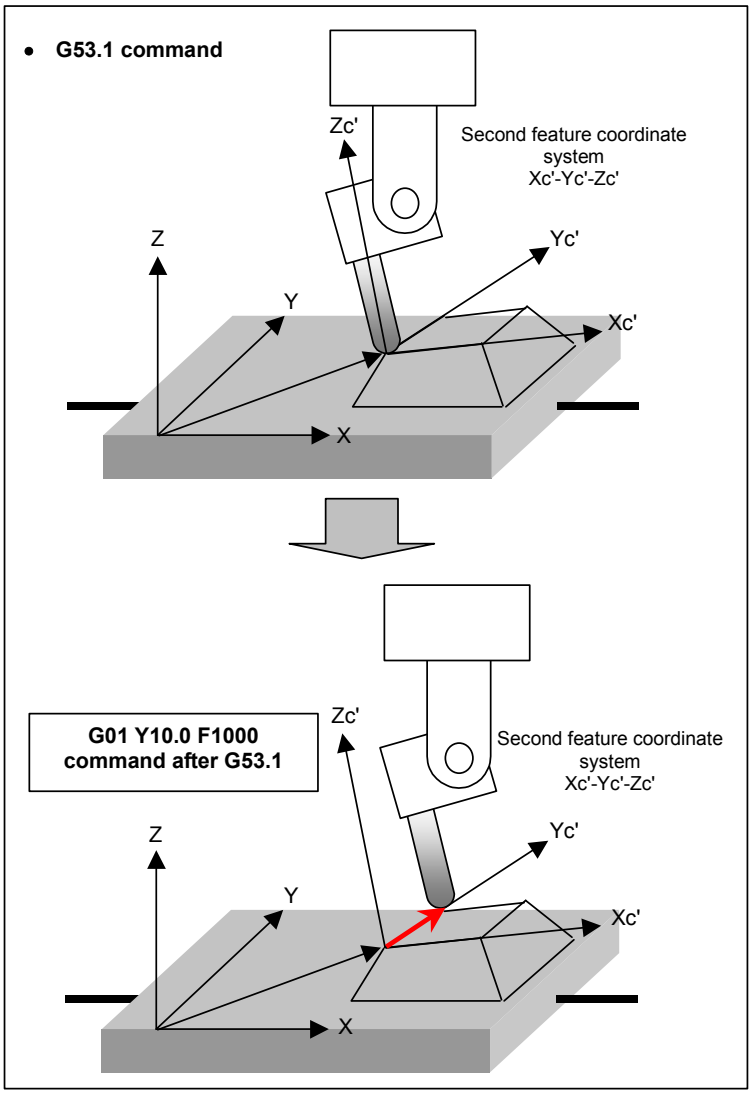


Fig. 22.6.3 (j) Resetting of the feature coordinate system

- Rotation direction of the table rotation axis

The composite type machine shown in Fig. 22.6.3 (i) is explained as an example.
Set parameter No.19684 to 1 if the rotation direction of the rotation table corresponding to the positive-direction move command is clockwise when viewed from the positive direction of the rotation center axis on which the table rotation axis rotates. If the rotation direction is counterclockwise, set parameter No.19684 to 0.
Let's take sample program 4 of O400 as an example, where the movement of the table is specified by G53.1.
If parameter No.19684 is set to 1, control is performed in such a way that the table is rotated to A-45.0.
If parameter No.19684 is set to 0, control is performed in such a way that the table is rotated to A45.0.
Example)

```
O400 (Sample Program4) ;
N1 G68.2 X100.0 Y100.0 Z0 I180.0 J45.0 K0 ;
N2 G53.1 ;
N3 . . . ;
```

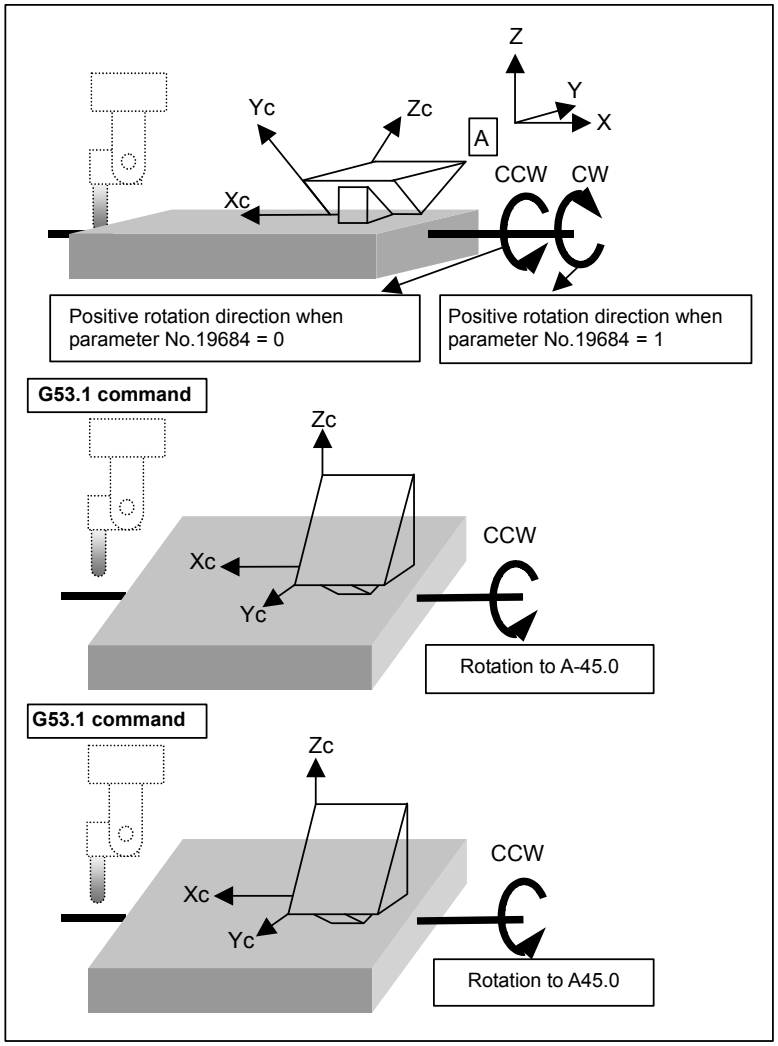


Fig. 22.6.3 (k) Rotation direction of the table rotation axis

- Table rotation type machine Basic operation

This function is also usable for a table rotation type machine with two table rotation axes.
The feature coordinate system Xc-Yc-Zc is set in the workpiece coordinate system based on the coordinate system origin shift (xo, yo, zo) and the Euler's angle.
Given the A-axis and C-axis shown in Fig. 22.6.3 (l), the A-axis and C-axis rotate until Zc comes in the X-Z plane and the tool axis is directed toward the +Z-axis direction of the feature coordinate system.

- **Tool axis direction control for a table rotation type machine**

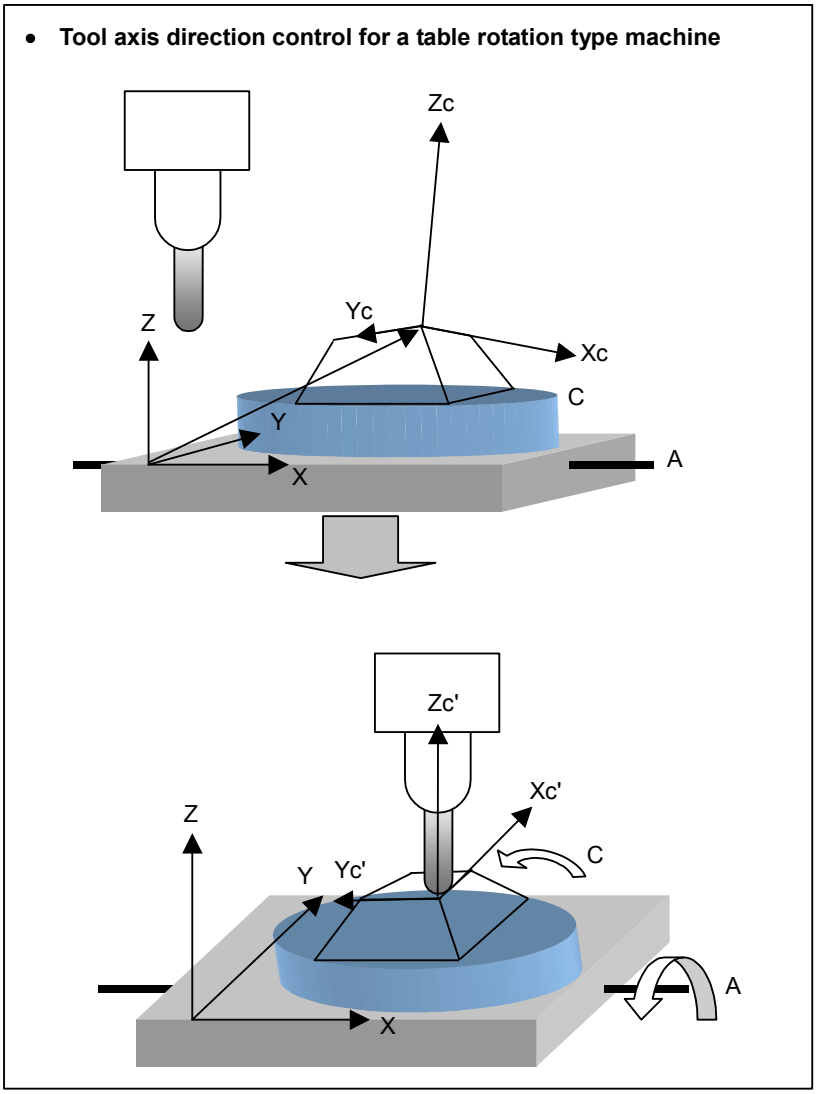


Fig. 22.6.3 (l) Table rotation type machine

- Feature coordinate system with the table rotated by G53.1 (tool axis direction control)

The table rotation type machine shown in Fig. 22.6.3 (l) is explained as an example.
If the table rotates by the tool axis direction control command (G53.1), the feature coordinate system (called the first feature coordinate system), which is set in the workpiece coordinate system by the tilted working plane indexing (G68.2), rotates as much as the table rotates.
The feature coordinate system that has rotated is called the second feature coordinate system.
Once G53.1 is specified, the subsequent machining commands are assumed to be specified in the second feature coordinate system. (See Fig. 22.6.3 (m).)
In the table rotation type machine, the specified feature coordinate system (the first feature coordinate system) may differ from the feature coordinate system to be used for machining (the second feature coordinate system).

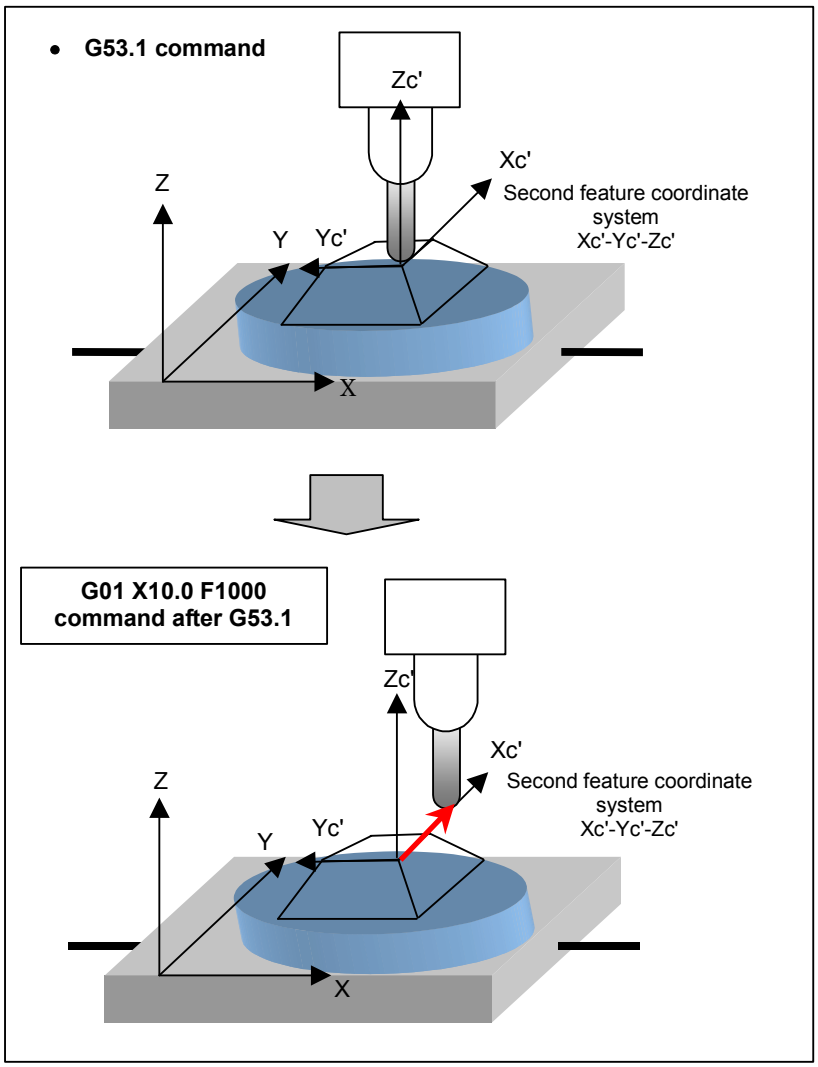


Fig. 22.6.3 (m) Resetting of the feature coordinate system

- Angle of the rotary axis

When tool axis direction control (G53.1) has been performed, more than two pairs of "computed angles" of the rotary axes usually exist.
The "computed angle" is the candidate angle at which the rotary axis is to be controlled in the tool axis direction specified by G53.1.
The "output angle" is determined from the "computed angle" based on the "output judgment conditions" described below.
When the upper and lower limits of the movement range of rotary axes are specified by parameters No. 19741 to No. 19744 at this time, a calculated angle that makes the two axes fall within the movement range is selected.

"Output judgment conditions"

Tool rotation type or table rotation type machine
<1> The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) moving angle is smaller. ↓ ↓ When the master axis moving angle is the same ↓
<2> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) moving angle is smaller. ↓ ↓ When the slave axis moving angle is the same ↓
<3> The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees). ↓ ↓ When the master axis angle is equally near to 0 degree ↓
<4> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).
Composite type machine
<1> The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) moving angle is smaller. ↓ ↓ When the table moving angle is the same ↓
<2> The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) moving angle is smaller. ↓ ↓ When the tool moving angle is the same ↓
<3> The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees). ↓ ↓ When the master axis angle is equally near to 0 degree ↓
<4> The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

The process of judging whether the moving angle is smaller or larger as the output judgement condition is called "movement judgement."

The "movement judgement" process is explained below.
When the "computed angle" is within the range between 0 and 360 degrees, it is called the "basic computed angle."
Usually, two pairs of "basic computed angles" exist.
For example, assume that the machine has rotary axis A (master) and rotary axis B (slave) and that there are two pairs of basic computed angles as follows:
(A $\theta 1$ degree; B $\phi 1$ degree)
(A $\theta 2$ degrees; B $\phi 2$ degrees) where $\theta 1 \leq \theta 2$ and $\phi 1 \leq \phi 2$.
The "computed angle" is obtained from either of the following expressions: "basic computed angle" + 360 degrees $\times$ N or "basic computed angle" - 360 degrees $\times$ N.
The current position of rotary axis A (master) is PA, and that of rotary axis B (slave) is 0 degree.
Based on the PA angle, the "movement judgement" process is done as follows.

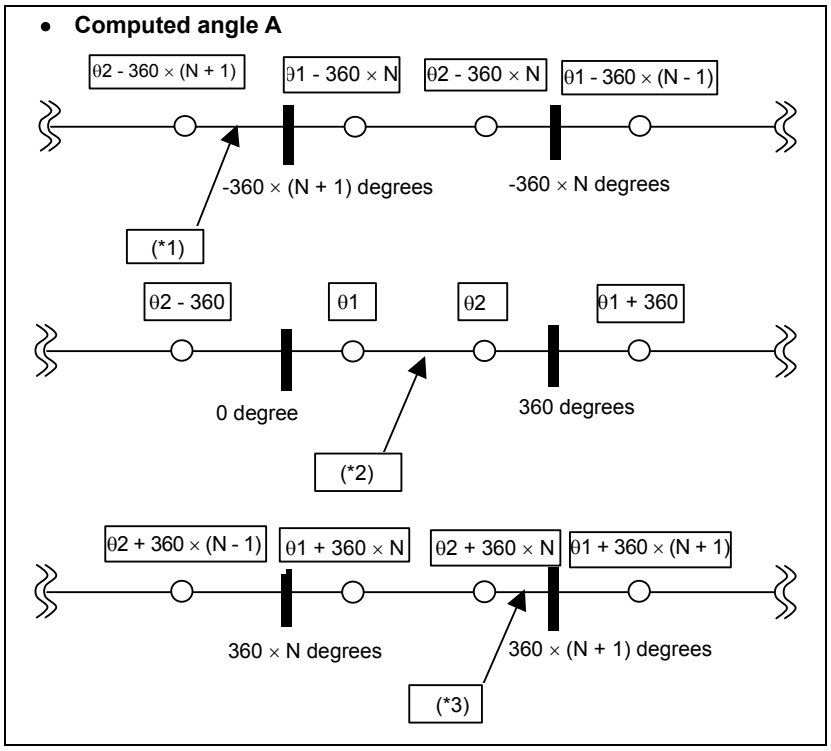


Fig. 22.6.3 (n) "Movement judgment"

When the PA angle is (*1):
The output angle is: (A θ2 - 360 × (N + 1) degrees; B ϕ2 degrees).
Namely, θ2 - 360 × (N + 1) degrees is adopted that is nearer to the computed angle of A, and ϕ2, which is the same group as θ2, is adopted as the computed angle of B.

When the PA angle is (*2):
The output angle is: (A θ1 degrees; B ϕ1 degrees).
Namely, θ1 degrees is adopted that is nearer to the computed angle of A, and ϕ1, which is the same group as θ1, is adopted as the computed angle of B.

When the PA angle is (*3):
The output angle is: (A θ2 + 360 × N degrees; B ϕ2 degrees).
Namely, θ2 + 360 × N degrees is adopted that is nearer to the computed angle of A, and ϕ2, which is the same group as θ2. is adopted as the computed angle of B.

When the moving angle of rotary axis A (master) is the same, a "movement judgement" is made for rotary axis B (slave) according to the "output judgment conditions."
If the "output angle" of rotary axis A is determined by the "movement judgement" for rotary axis A, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis B.
Similarly, if the "output angle" of rotary axis B is determined by the "movement judgement" for rotary axis B, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis A.

⚠ CAUTION
1 To use the rotary axis roll-over function, set parameter No. 1260 (amount of rotary axis movement per rotation) to 360 degrees.
2 A stroke limit before movement is applied to the rotary axis subject to tool axis direction control.

⚠ CAUTION

3 If the setting of the lower limit (parameters No. 19742 and No. 19744) is greater than that of the upper limit (parameters No. 19741 and No. 19743), alarm PS5459 is issued.
4 If there is no calculated angle that falls within the movement range because the movement range is too small, alarm PS5459 is issued.
5 When the parameters that specify the upper limit and lower limit of the movement range are set to 0, no movement range is assumed to be specified.

The "output angle" is explained below using a tool rotation type machine as an example.
This example illustrates a machine having a "BC type tool axis Z."

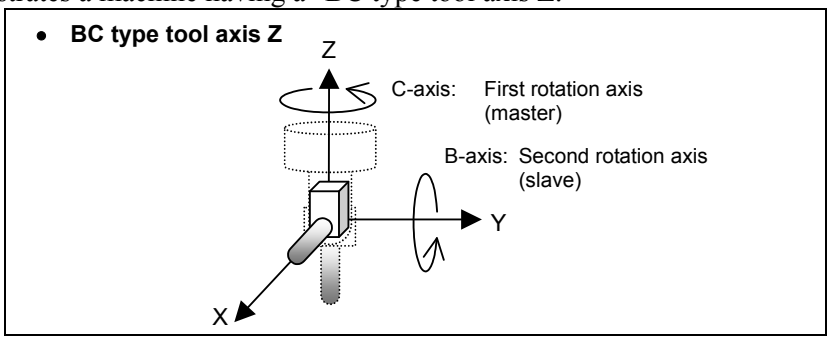


Fig. 22.6.3 (o) BC type tool axis Z

The following two pairs of "computed basic angles" exist that direct the tool axis toward the +X-axis direction.
(B 90 degrees; C 180 degrees)
(B 270 degrees; C 0 degree)

<1> When the current rotary axis angles are (B -70 degrees; C 30 degrees)
The "output angles" are (B -90 degrees; C 0 degree).
0 degree is adopted because it is nearer to the current position (30 degrees) of the C-axis that is the master axis. For the B-axis, 270 degrees is adopted which is the same group. However, this is changed to -90 degrees (270 degrees - 360 degrees) which is the nearest to the current position of the B-axis (-70 degrees).
<2> When the current rotary axis angles are (B 80 degrees; C 500 degrees)
The "output angles" are (B 90 degrees; C 540 degrees).
540 degrees (180 degrees + 360 degrees) is adopted because it is nearer to the current position (500 degrees) of the C-axis that is the master axis. For the B-axis, 90 degrees is adopted which is the same group.
<3> When the current rotary axis angles are (B 60 degrees; C 90 degrees)
The "output angles" are (B 90 degrees; C 180 degrees).
Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. 90 degrees is adopted because it is nearer to the current position (60 degrees) of the B-axis that is the slave axis. For the C-axis, 180 degrees is adopted which is the same group.
<4> When the current rotary axis angles are (B 180 degrees; C 90 degrees)
The "output angles" are (B 270 degrees; C 0 degree).
Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. In this case, however, the two candidates are also equally near to the current position of the B-axis (180 degrees). Therefore, the candidate is adopted in which the C-axis (master axis) is nearer to 0 degree.
That is, the pair is adopted whose C-axis angle is 0 degree and whose B-axis angle is 270 degrees.

When the slave axis angle is 0 degree, the direction of the tool axis becomes fixed regardless of the master axis angle.
In that case, the master axis does not move from the current angle.

An explanation is shown below using a machine having a "BC type tool axis Z" as an example.

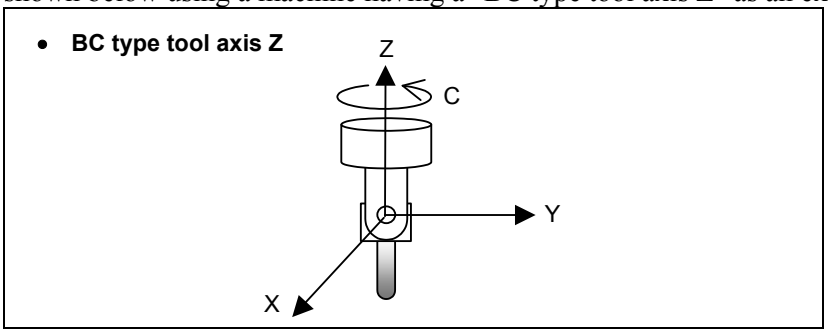


Fig. 22.6.3 (p) BC type tool axis Z

When the current rotary axis angles are (B 45 degrees; C 90 degrees), the "output angles" are (B 0 degree; C 90 degrees).

- Specification for rotary axes

The tilted working plane indexing assumes that positioning of a rotary axis is performed by tool axis direction control (G53.1) and then machining is performed without the rotary axis being moved.
When movement is made with the rotary axis specified directly, the movement of the rotary axis is not considered in the movement of the linear axis. Do not specify the movement of the rotary axis during execution of the tilted working plane indexing to ensure correct machining.

Second Rotation Axis Control in Tool Axis Direction Control Where the End Point is a Singular Point

When the end point of tool axis direction control (G53.1/G53.6) during execution of the tilted working plane indexing is a singular point, the second rotation axis is controlled so that the direction of the second feature coordinate system matches that of the workpiece coordinate system.
To enable this function, set bit 4 (CFW) of parameter No. 11221 to 1.

When bit 4 (CFW) of parameter No. 11221 is set to 0, if the end point of tool axis direction control (G53.1/G53.6) is a singular point, the second rotation axis does not operate and only the first rotation axis turns. As a result, the X- and Y-directions of the feature coordinate system depend on the second rotation axis immediately before tool axis direction control. (Fig. 22.6.3 (a), Fig. 22.6.3 (b))

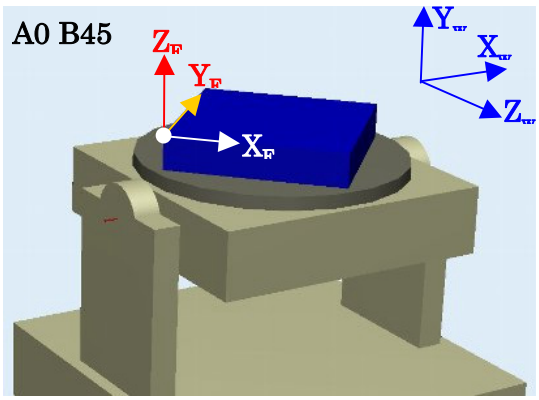


Fig. 22.6.3 (a) Before G53.1

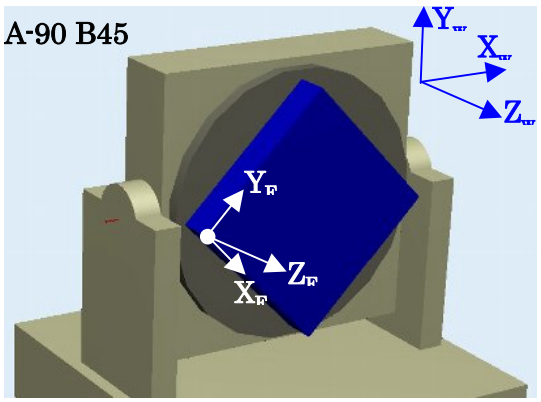


Fig. 22.6.3 (b) After G53.1
The second rotation axis (B-axis) does not turn.

When this function is enabled (bit 4 (CFW) of parameter No. 11221 = 1), the second rotation axis is controlled so that the direction of the second feature coordinate system matches that of the workpiece coordinate system. (Fig. 22.6.3 (c),Fig. 22.6.3 (d))

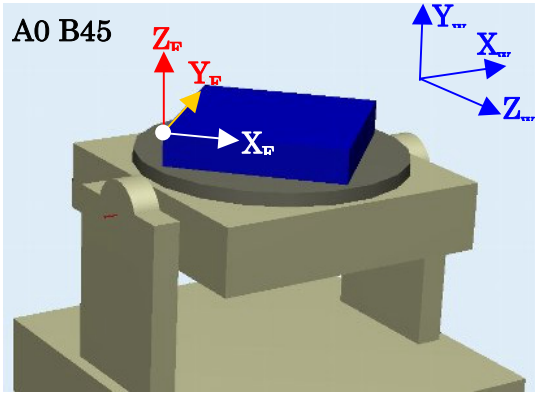


Fig. 22.6.3 (c) Before G53.1

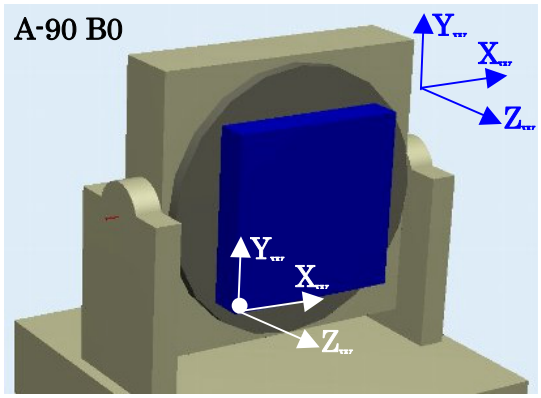


Fig. 22.6.3 (d) After G53.1

The second rotation axis (B-axis) moves so that the direction of the second feature coordinate system matches that of the workpiece coordinate system.

22.6.3.2 Tool center point retention type tool axis direction control

In tool axis direction control after the tilted working plane indexing, tool center point retention type tool axis direction control can be specified. In tool center point retention type tool axis direction control, the tool moves so that it is perpendicular to the tilted plane while retaining the tool center point on the workpiece.

By specifying the distance between the tool center point to the rotation center with R, it is possible to move the tool so that it is perpendicular to the titled plane while retaining the rotation center shifted from the tool center point.

Format

M

G53.6 (H_) (R_) ; Tool center point retention type tool axis direction control

H : Tool compensation number
R : Distance from the tool center point to the rotation center
(Specify R as a radius value.)

G53.6 is a one shot G code.

To use tool center point retention type tool axis direction control, a tool length offset number is required. If no H command is specified, the H code is regarded as modal information.

To shift the rotation center from the tool center point, enter the distance from the tool center point to the rotation center, using the R command. If no R command is specified, the distance is regarded as R0, and the tool moves while retaining the tool center position on the workpiece so that it is perpendicular to the tilted plane.

T

G53.6 (D_) (R_) ; Tool center point retention type tool axis direction control

D : Tool compensation number
R : Distance from the tool center point to the rotation center
(Specify R as a radius value.)

To use tool center point retention type tool axis direction control on a T series machine, enable the extended tool selection function (set bit 3 (TCT) of parameter No. 5040 = 1).
Use D as the tool length offset number.
Other restrictions are the same as those on the M series.

⚠ CAUTION

1 Specify tool center point retention type tool axis direction control (G53.6) in tool length compensation cancel mode (G49). If it is specified in a mode other than G49 mode, alarm PS5458 is issued.
2 If G68.2 is not specified before G53.6 is specified, alarm PS5458 is issued.
3 Specify G53.6 independently. If an axis movement command is specified in the same block, alarm PS5458 is issued.
4 The tool center point retention type tool axis direction control (G53.6) block becomes a block that suppresses buffering.
5 If tool center point retention type tool axis direction control does not have an H command (D command for the T series), and the modal information for H (D) is 0, alarm PS5458 is issued.
6 For the feedrate, the movement speed of the rotation axis is applied. During rapid traverse, it is regarded as the maximum rapid traverse rate, and as the specified speed during cutting feed.
7 Specify tool center point retention type tool axis direction control (G53.6) in cutter compensation cancel mode (G40). If it is specified in a mode other than G40 mode, alarm PS5458 is issued.
8 Specify tool center point retention type tool axis direction control (G53.6) in either G00 or G01 mode.
9 Specify tool center point retention type tool axis direction control (G53.6) with feed per minute or feed per revolution.
10 Do not specify tool center point retention type tool axis direction control (G53.6) while in workpiece setting error compensation mode. Otherwise, alarm PS5458 is issued.
11 Do not perform manual intervention during tool center point retention type tool axis direction control (G53.6). Otherwise, alarm PS5458 is issued.
12 If the R command is specified, and if an exceedingly large value is input as R, alarm PS143 may be issued.
13 If a movement range is set for a roll over axis (parameters Nos. 19741 to 19744 $\neq$ 0), the roll over function is disabled during tool center point retention type tool axis direction control (G53.6).

Example

Tool center point retention type tool axis direction control with R not specified
Fig. 22.6.3 (a) and Fig. 22.6.3 (b) show tool center point retention type tool axis direction control with R not specified. The tool moves so that it is perpendicular to the tilted plane while retaining the tool center point on the workpiece.
This function can be used by specifying G53.6 and specifying a tool length offset number with H (D for the T series). (If H (D) has the modal information for the currently used tool, this function can be used without specifying H (D).)

```
O0002(TCP-HOLD-TYPE)
G00 B0 C0
G5.1 Q1
G68.2 X0 Y0 Z0 I90.0 J45.0 K0
```

G53.6 H1 (G53.6 D1 for the T series)

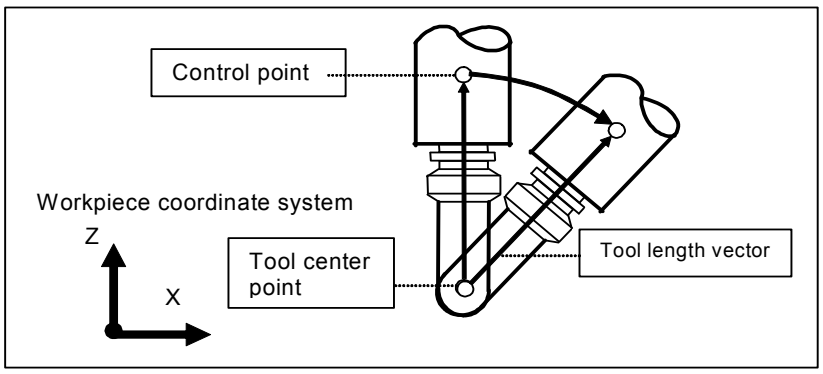


Fig. 22.6.3 (a) Operation of tool center point retention type tool axis direction control (tool rotation type)

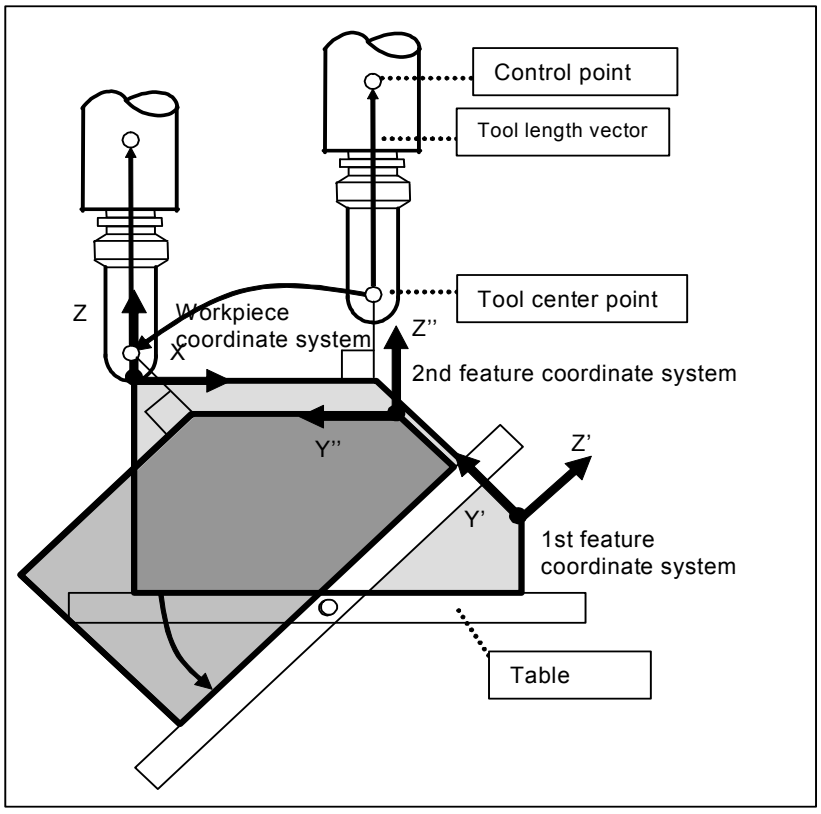


Fig. 22.6.3 (b) Operation of tool center point retention type tool axis direction control (table rotation type)

Rotation center compensation with the tool center point retention type
In tool center point retention type tool axis direction control, the rotation center can be shifted from the tool center point by specifying R.
Fig. 22.6.3 (c) and Fig. 22.6.3 (d) show cases in which the rotation center is shifted by specifying the distance from the tool center point to the workpiece with R.
By specifying this, the tool moves so that it is perpendicular to the titled plane while retaining the rotation center on the workpiece.
This function can be used by specifying G53.6 and specifying a tool length offset number with H (D for the T series) and specifying the distance from the tool center point to the rotation center with R.

O0003(CENTER-OF-ROTATION-HOLD-TYPE)
G00 B0 C0
G5.1 Q1
G68.2 X0 Y0 Z0 I90.0 J45.0 K0

G53.6 H1 R200.0 (G53.6 D1 R200.0 for the T series)

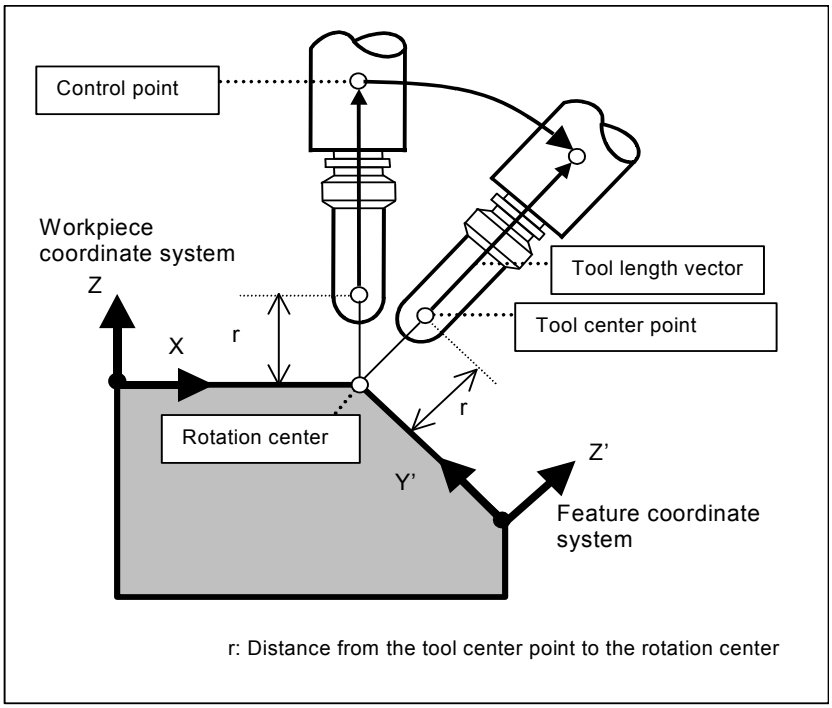


Fig. 22.6.3 (c) Operation of rotation center compensation with tool center point retention type tool axis direction control (tool rotation type)

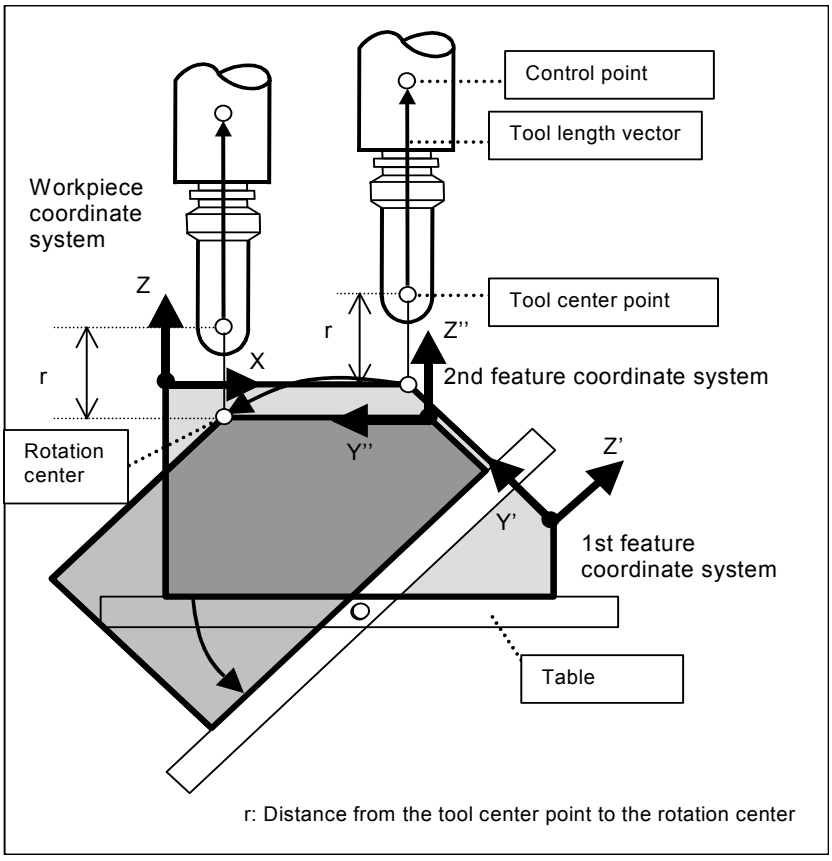


Fig. 22.6.3 (d) Operation of rotation center compensation with tool center point retention type tool axis direction control (table rotation type)

22.6.4 Tilted Working Plane Indexing in Tool Length Compensation

Overview

In tool length compensation (G43), G68.2/G68.4 (tilted working plane indexing) and G53.1 (tool axis direction control)/G53.6 (tool center point retention type tool axis direction control) can be specified.
Accordingly, the G68.2/G68.4 and G53.1/G53.6 commands can be used without canceling tool length compensation.

Explanation

- G68.2/G68.4 command in tool length compensation

The G68.2/G68.4 command can be executed in tool length compensation.
Absolute coordinates after the G68.2/G68.4 command are based on the position of the tool center point on the feature coordinate system.
When the tilted working plane indexing is executed with the tool or table tilted on the rotation axis, absolute coordinates are based on the position of the tool center point with the position of the rotation axis considered.
Accordingly, machining is allowed even when the tool axis direction is not the Z-axis direction on the feature coordinate system.

Example of operation 1

```
N10 G69 ;
N20 G54 G43 H1 X0 Y0 Z0 ;
N30 G68.2 X_ Y_ Z_ I90. 0J-30.0 K-90.0 ; (rotation by -30 degrees about the Y-axis)
N40 X100.0 Y0 Z0 ;
```

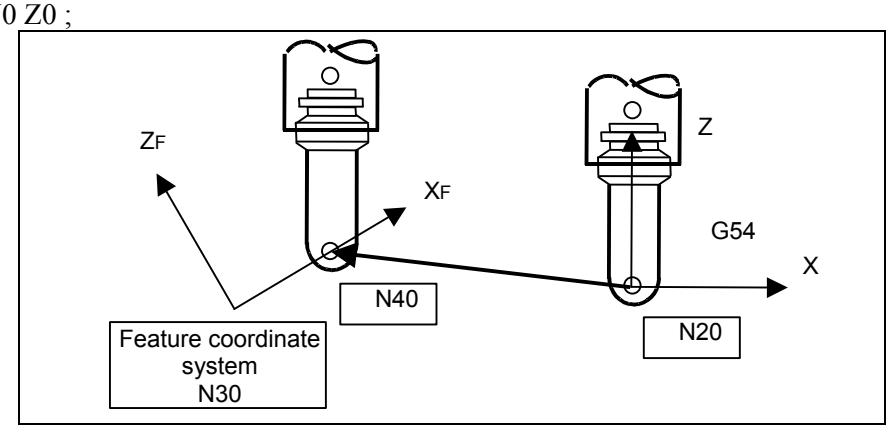


Fig. 22.6.4 (a) Example of operation 1

Example of operation 2

```
N10 G69 ;
N20 G54 G43 H1 X0 Y0 Z0 B0 ;
N30 B20.0 ;
N40 G68.2 X_ Y_ Z_ I90.0 J-30.0 K-90.0 ; (rotation by -30 degrees about the Y-axis)
N50 X100.0 Y0 Z0 ;
```

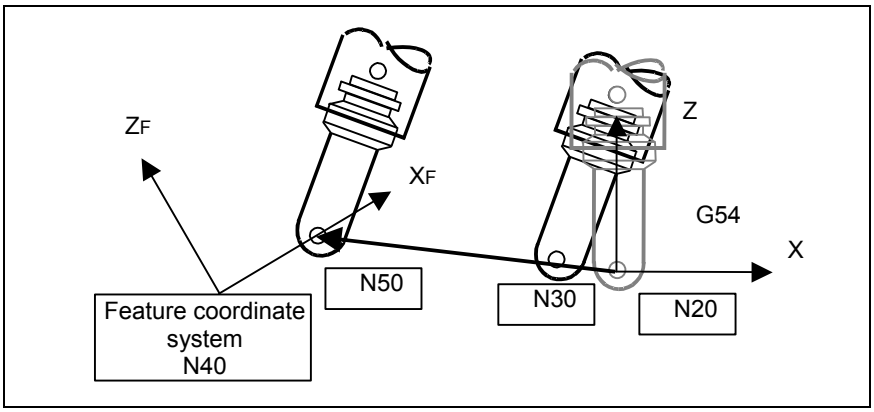


Fig. 22.6.4 (b) Example of operation 2 (tool rotation type)

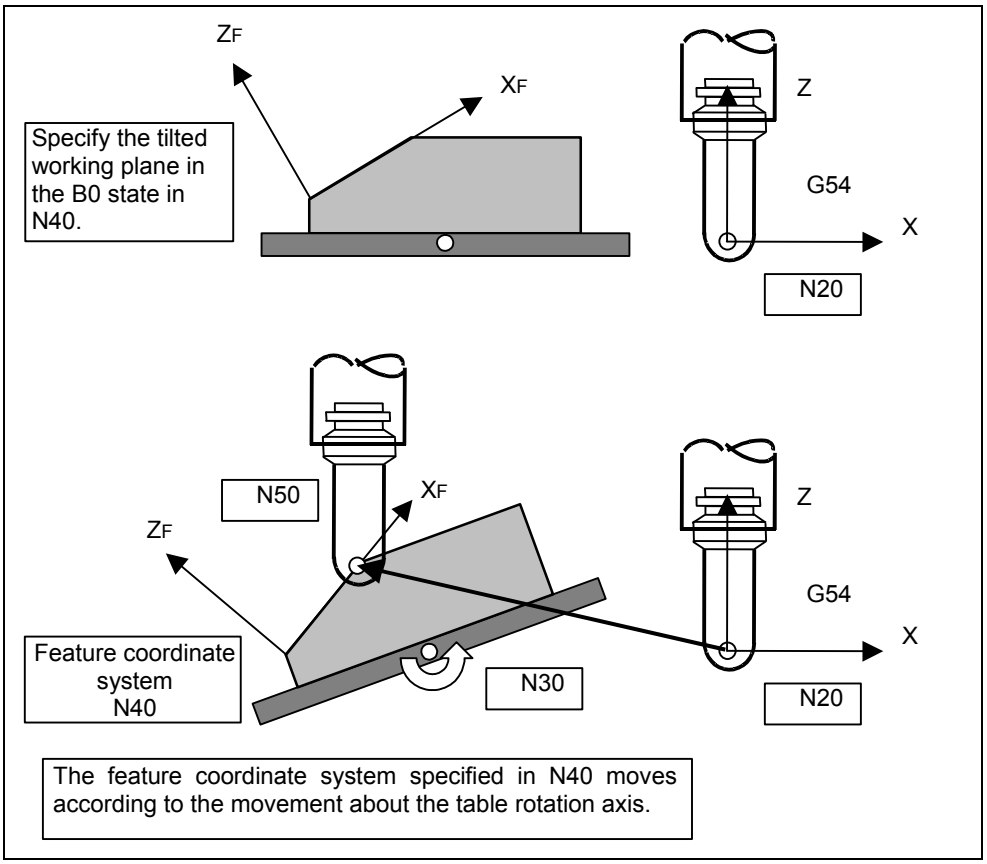


Fig. 22.6.4 (c) Example of operation 2 (table rotation type)

Example of operation 3

N30 **G43** X0 Y0 Z0 B20.0 ;
N40 **G68.2** X_ Y_ Z_ I90.0 J-30.0 K-90.0 ; (rotation by -30 degrees about the Y-axis)
N50 X100.0 Y0 Z0 ;
N60 B-20.0
N70 **G68.4** X_ Y_ Z_ I90.0 J40.0 K-90.0 ; (incremental multiple command: rotation by 40 degrees about the Y-axis)
N80 X100.0 Y0 Z0 ;

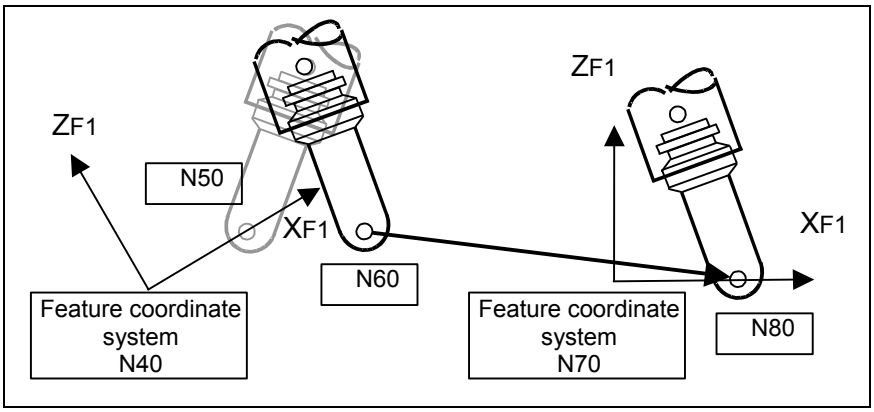


Fig. 22.6.4 (d) Example of operation 3 (tool rotation type)

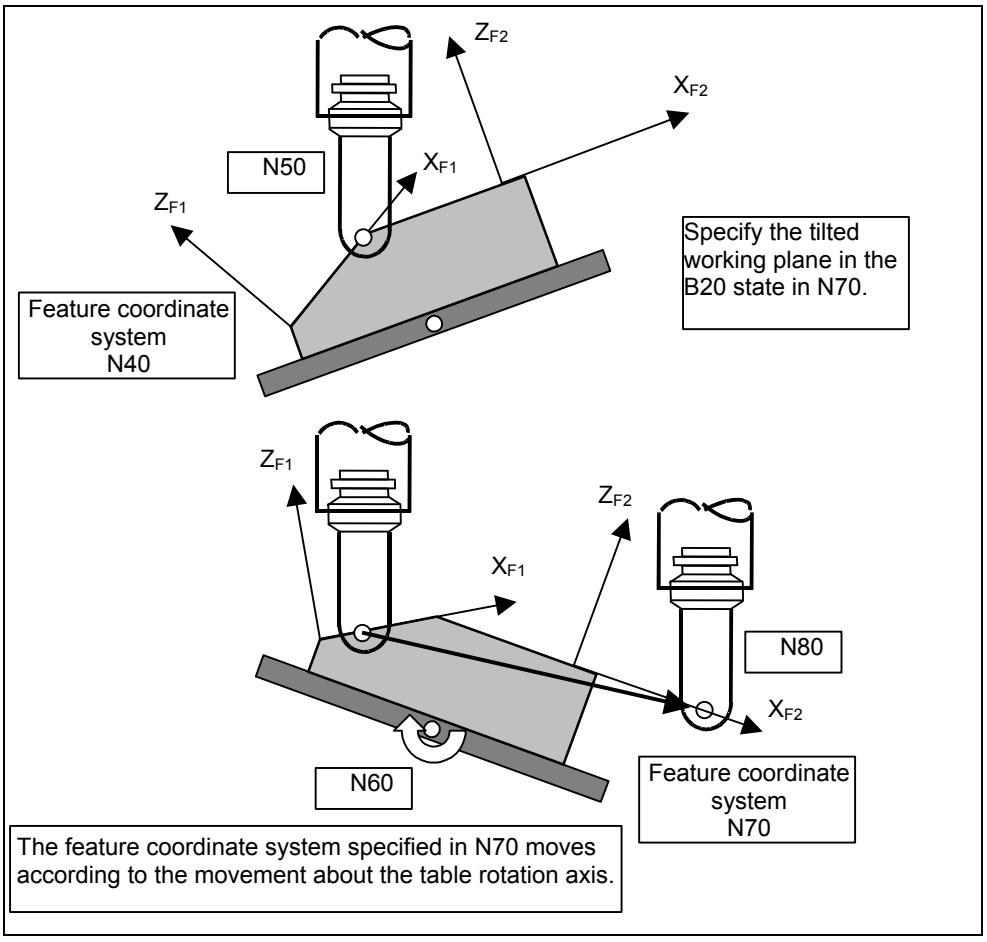


Fig. 22.6.4 (e) Example of operation 3 (table rotation type)

G53.1 command in tool length compensation

The G53.1 command can be executed in tool length compensation.

G53.1 operation in tool length compensation is performed in the same way as in the tool length compensation cancel mode.

Absolute coordinates after the G53.1 command are based on the position of the tool center point on the feature coordinate system after the G53.1 command is specified.

Example of operation 4

N10 G54 **G43** H1 X_ Y_ Z_ ;
N20 **G68.2** X_ Y_ Z_ I90.0 J-30.0 K-90.0 ; (rotation by -30 degrees about the Y-axis)
N30 **G53.1** ;

N40 X100.0 Y0 Z0 ;

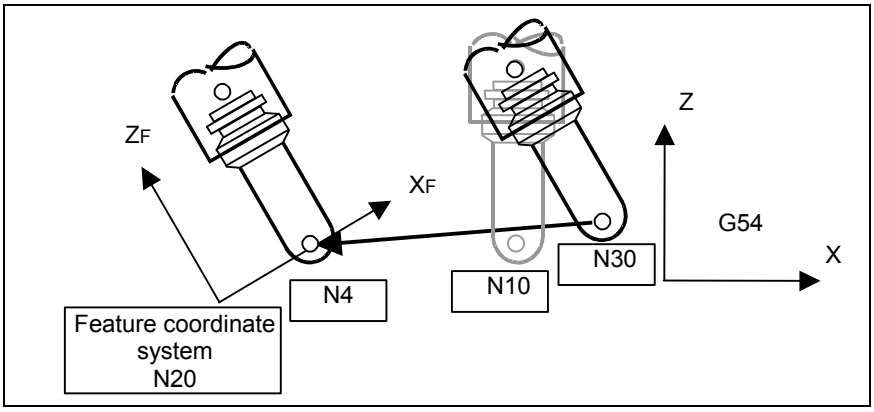


Fig. 22.6.4 (f) Example of operation 4 (tool rotation type)

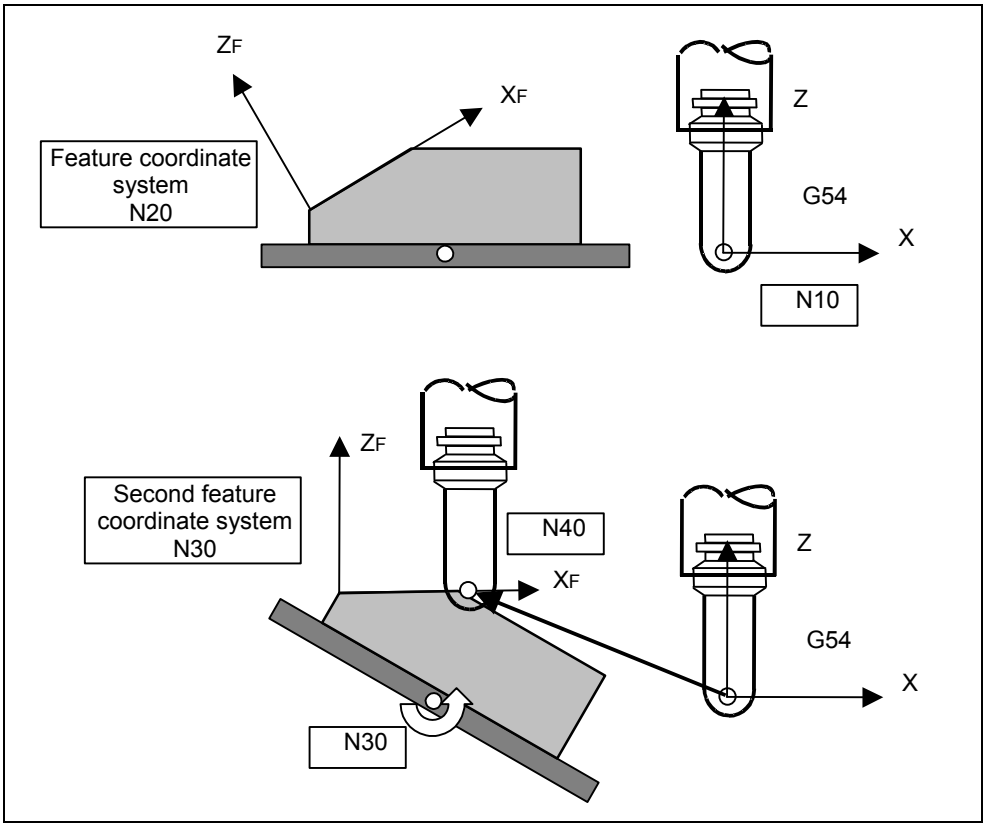


Fig. 22.6.4 (g) Example of operation 4 (table rotation type)

G53.6 command in tool length compensation

The G53.6 command can be executed in tool length compensation.

G53.6 operation in tool length compensation is performed in the same way as in the tool length compensation cancel mode.

G69 command in tool length compensation

The G69 command can be executed in tool length compensation.

After G69 operation in tool length compensation, the tool length compensation vector represents the Z direction of the workpiece coordinate system regardless of the position of the rotation axis.

> ⚠ **CAUTION**
> Any tilted working plane indexing in tool length compensation cannot be used in workpiece setting error compensation.

Restrictions

- Basic restrictions

The restrictions imposed on 3-dimensional coordinate conversion also apply to the tilted working plane indexing.

- Increment system

The same increment system must be used for the basic three axes used by this function.

- Rapid traverse command

Linear rapid traverse (bit 1 (LRP) of parameter No. 1401 = 1) must be set for the rapid traverse command.

- 3-dimensional coordinate system conversion

If an attempt is made to set a new coordinate system by performing 3-dimensional coordinate conversion in a feature coordinate system, an alarm is also raised.

- Positioning in the machine coordinate system

Positioning commands in the machine coordinate system such as G28, G30, and G53 operate in the machine coordinate system rather than the feature coordinate system.

- External mirror image

If an attempt is made to use this function and an external mirror image function simultaneously, this function takes effect before the external mirror image function.

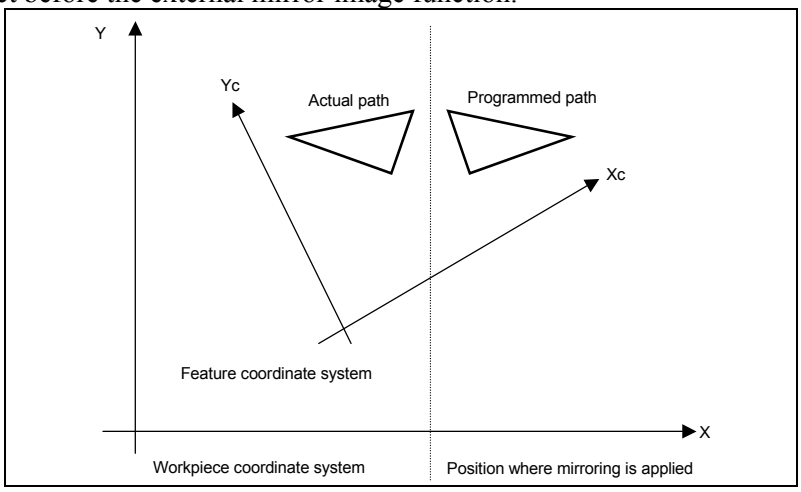


Fig. 22.6.4 (h)

- Tool center point retention type tool axis direction control

Performing a reset during tool center point retention type tool axis direction control results in the operation to be performed when the following parameters are set regardless of whether the reset is performed in the foreground or background. Thus, do not use the background during tool center point retention type tool axis direction control.

(1) Bit 2 (D3R) of parameter No. 5400 = 0: Clears the tilted working plane indexing.
(2) Bit 6 (CLR) of parameter No. 3402 = 1: Clears with a reset.
(3) Bit 0 (C08) of parameter No. 3407 = 0: Clears the G code in group 08.
(4) Bit 7 (C23) of parameter No. 3408 = 0: Clears the G code in group 23.
(5) Bit 7 (CFH) of parameter No. 3409 = 0: Clears the F, H, D, and T codes.
(6) Bit 6 (LVK) of parameter No. 5003 = 0: Clears the tool length compensation vector.

- Relationships with other modal commands

G41, G42, and G40 (cutter compensation), G43, G49 (tool length compensation), G51.1 and G50.1 (programmable mirror image), and canned cycle commands must have nesting relationships with G68.2. In other words, first issue G68.2 when the modes mentioned above are off, turn the modes on and off, then issue G69.

- Manual reference position return

Manual reference position return in the tilted working plane indexing mode results in alarm PS5324. If you want to perform manual reference position return, cancel the tilted working plane indexing mode.

- Hypothetical axis of a table rotation axis

When a table rotation axis is set as a hypothetical axis, tilted working plane indexing is performed on the assumption that the table rotation axis is at 0 degrees.

- Specifiable G codes

The G codes that can be specified in the tilted working plane indexing mode are listed below.
Specifying a G code other than these codes results in alarm PS5462.

(1) Positioning (G00)
(2) Linear interpolation (G01)
(3) Circular interpolation / helical interpolation (G02/G03)
(4) Dwell (G04)
(5) Exact stop(G09)
(6) Programmable data input (G10)
(7) Programmable data input mode cancel (G11)
(8) Plane selection (G17/G18/G19)
(9) Automatic return to reference position (G28)
(10) Movement from reference position (G29)
(11) 2nd, 3rd and 4th reference position return (G30)
(12) Skip function (G31)
(13) Cutter compensation : cancel (G40)
(14) Tool radius or tool nose radius compensation / 3-dimensional cutter compensation (G41/G42)
(15) Tool length compensation + (G43)
(16) Tool length compensation - (G44)
(17) Tool center point control (G43.4/G43.5)
(18) Tool length compensation cancel (G49,G49.1)
(19) Machine coordinate system setting (G53)
(20) Tool axis direction control (G53.1)
(21) Workpiece coordinate system setting (G54 to G59, G54.1)
(22) Exact stop (G61)
(23) Automatic corner override (G62)
(24) Tapping mode (G63)
(25) Cutting mode (G64)
(26) Macro call (G65)
(27) Macro modal call A (G66)
(28) Macro modal call B (G66.1)
(29) Macro modal call A/B cancel (G67)
(30) Absolute programming (G90)
(31) Incremental programming (G91)
(32) Canned cycle for drilling (G73, G74, G76, G80 to G89)
(33) Constant surface speed control (G96)
(34) Constant surface speed control cancel (G97)
(35) Canned cycle : return to initial level (G98)
(36) Canned cycle : return to R point level (G99)

M

(1) Programmable mirror image (G50.1/G51.1)
(2) Coordinate system rotation cancel or 3-dimensional coordinate system conversion mode off (G69)
(3) Feed per minute (G94)
(4) Feed per revolution (G94)

T

(1) Coordinate system rotation cancel or 3-dimensional coordinate system conversion mode off (G69.1)
(2) Feed per minute (G98 (G94))
(3) Feed per revolution (G99 (G94))

- Modal G codes that allow specification of a tilted working plane indexing

A tilted working planeindexing can be specified in the modal G code states listed below.
In a modal state other than the following modal G codes, specifying the tilted working plane indexing results in alarm PS5462:

(1) Modal G codes included in "Specifiable G codes" described previously (However, G41/G42 are excluded.)
(2) Polar coordinate interpolation mode cancel (G13.1)
(3) Polar coordinates command cancel (G15)
(4) Input in inch (G20 (G70))
(5) Input in mm (G21 (G71))
(6) Stored stroke check function (G22/G23)
(7) Scaling cancel (G50)
(8) Polygon turning cancel (G50.2)
(9) Workpiece coordinate system setting (G54 to G59)
(10) Workpiece setting error compensation (G54.4)
(11) Exact stop mode (G61)
(12) Automatic corner override (G62)
(13) Tapping mode (G63)
(14) Cutting mode (G64)
(15) Inverse time feed (G93)
(16) Constant surface speed control cancel (G97)

M

(1) Polar coordinate interpolation mode cancel (G113)

T

(1) Programmable mirror image cancel (G50.1)
(2) Mirror image for double turret off/balanced cutting mode cancel (G69)

22.7 INCLINED ROTARY AXIS CONTROL

Overview

The conventional tilted working plane indexing / tool center point control function / 3-dimensional cutter compensation / 3-dimensional manual feed can be used only for those machines whose tool rotation axis or table rotation axis is parallel to the basic axis of the basic coordinate system. (See Fig. 22.7 (a).)
Inclined rotary axis control is intended to allow the tilted working plane indexing / tool center point control function / 3-dimensional cutter compensation / 3-dimensional manual handle feed to be applied even when the tool rotation axis or table rotation axis is inclined with respect to the X-Z plane, a Y-Z plane or a Z-X plane of the machine coordinate system.

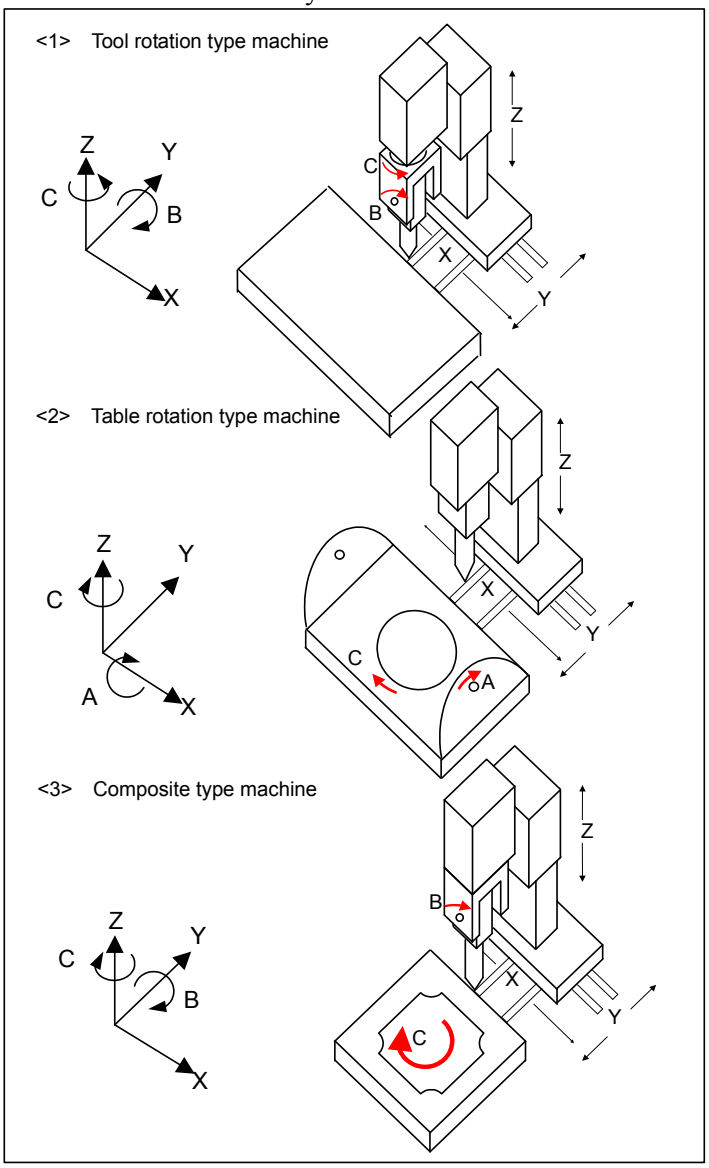


Fig. 22.7 (a) Three types of 5-axis machine

An example of a tool rotation type machine is explained below. (See Fig. 22.7 (b).)
The machine shown in Fig. 22.7 (b) has rotary axis B (master) that turns around the Y-axis and rotary axis C (slave) whose Y-axis is inclined at an angle of 45 degrees on the Y-Z plane.
The tilted working plane indexing / tool center point control function / 3-dimensional cutter compensation / 3-dimensional manual feed can be used even for the machine configuration shown in Fig. 22.7 (b).

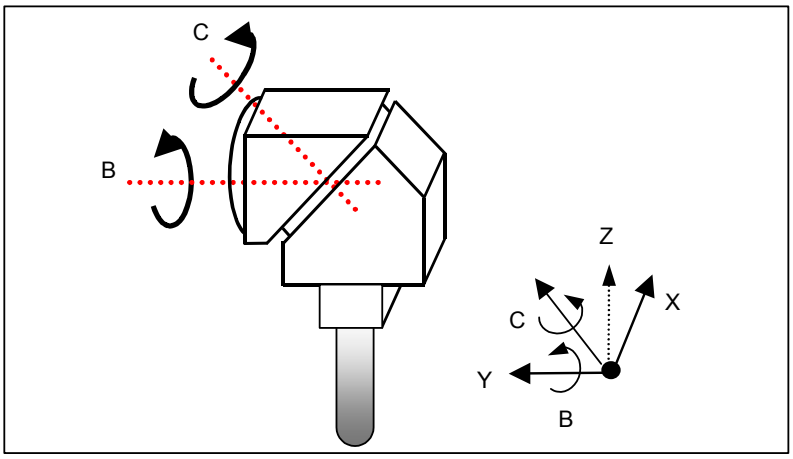


Fig. 22.7 (b) Tool rotation type machine

An example of a table rotation type machine is explained below. (See Fig. 22.7 (c).)
The machine shown in Fig. 22.7 (c) has rotary axis B (master) whose Y-axis is inclined at an angle of -45 degrees on the Y-Z plane and rotary axis C (slave) that turns around the Z-axis.
The tilted working plane indexing / tool center point control function / 3-dimensional cutter compensation / 3-dimensional manual feed can be used even for the machine configuration shown in Fig. 22.7 (c).

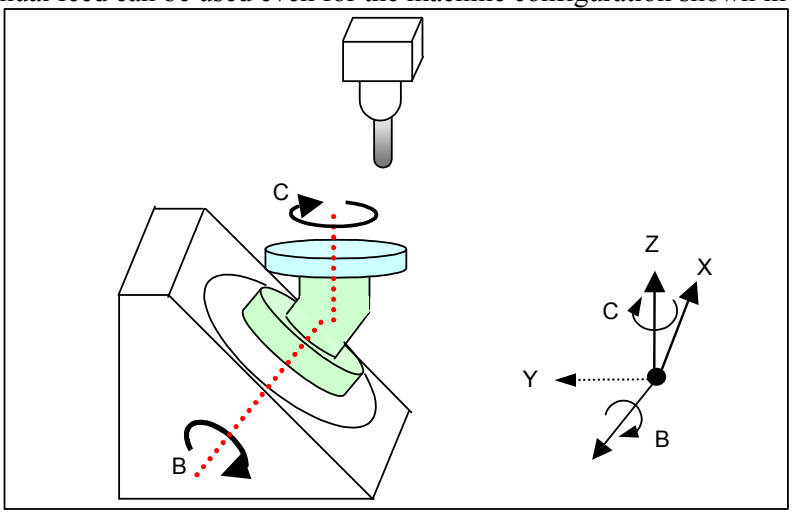


Fig. 22.7 (c) Table rotation type machine

An example of a composite type machine is explained below. (See Fig. 22.7 (d).)
The machine shown in Fig. 22.7 (d) has table rotation axis B whose Y-axis is inclined at an angle of -45 degrees on the Y-Z plane and tool rotation axis C that turns around the Z-axis.
The tilted working plane indexing / tool center point control function / 3-dimensional cutter compensation / 3-dimensional manual feed can be used even for the machine configuration shown in Fig. 22.7 (d).

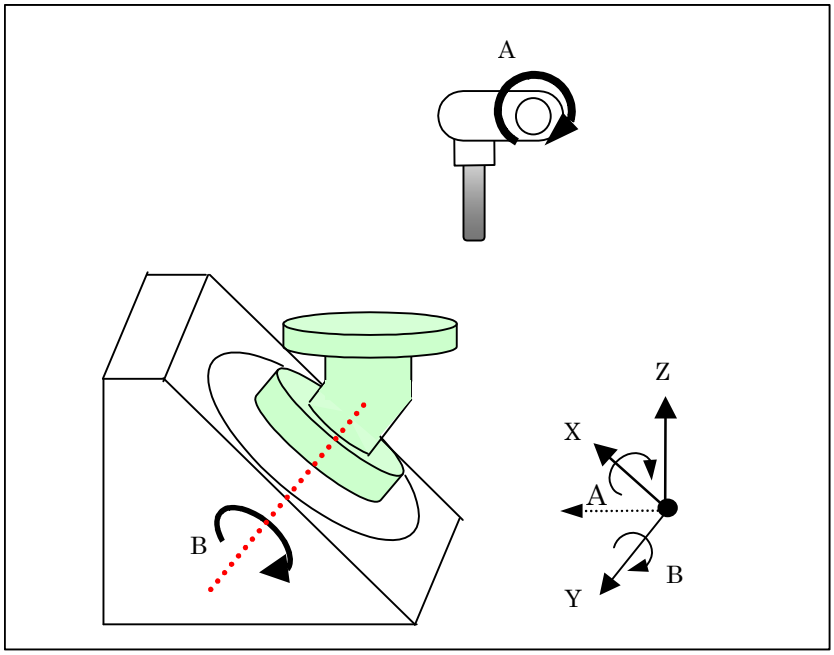


Fig. 22.7 (d) Composite type machine

Format and operation

The operation of the tilted working plane indexing / tool center point control function / 3-dimensional cutter compensation / 3-dimensional manual feed during the inclined rotary axis control is similar to the operation performed when the inclined rotary axis control is not in use.
For details, see the description of each function.

Limitation

The operation of the tilted working plane indexing / tool center point control function / 3-dimensional cutter compensation / 3-dimensional manual feed during the inclined rotary axis control is similar to the operation performed when the inclined rotary axis control is not in use.
For details, see the description of each function.

22.8 3-DIMENSIONAL CUTTER COMPENSATION

Overview

For machines having multiple rotary axes for freely controlling the orientation of a tool axis, this function calculates a tool vector from the positions of these rotary axes. The function then calculates a compensation vector in a plane (compensation plane) perpendicular to the tool vector and performs 3-dimensional cutter compensation.
The compensation is performed on the block end point.

- Machine configuration

This function is applicable to the following machine configurations:
<1> Tool rotation type machine controlled with two tool rotation axes
<2> Table rotation type machine controlled with two table rotation axes
<3> Composite type machine controlled with one tool rotation axis and one table rotation axis

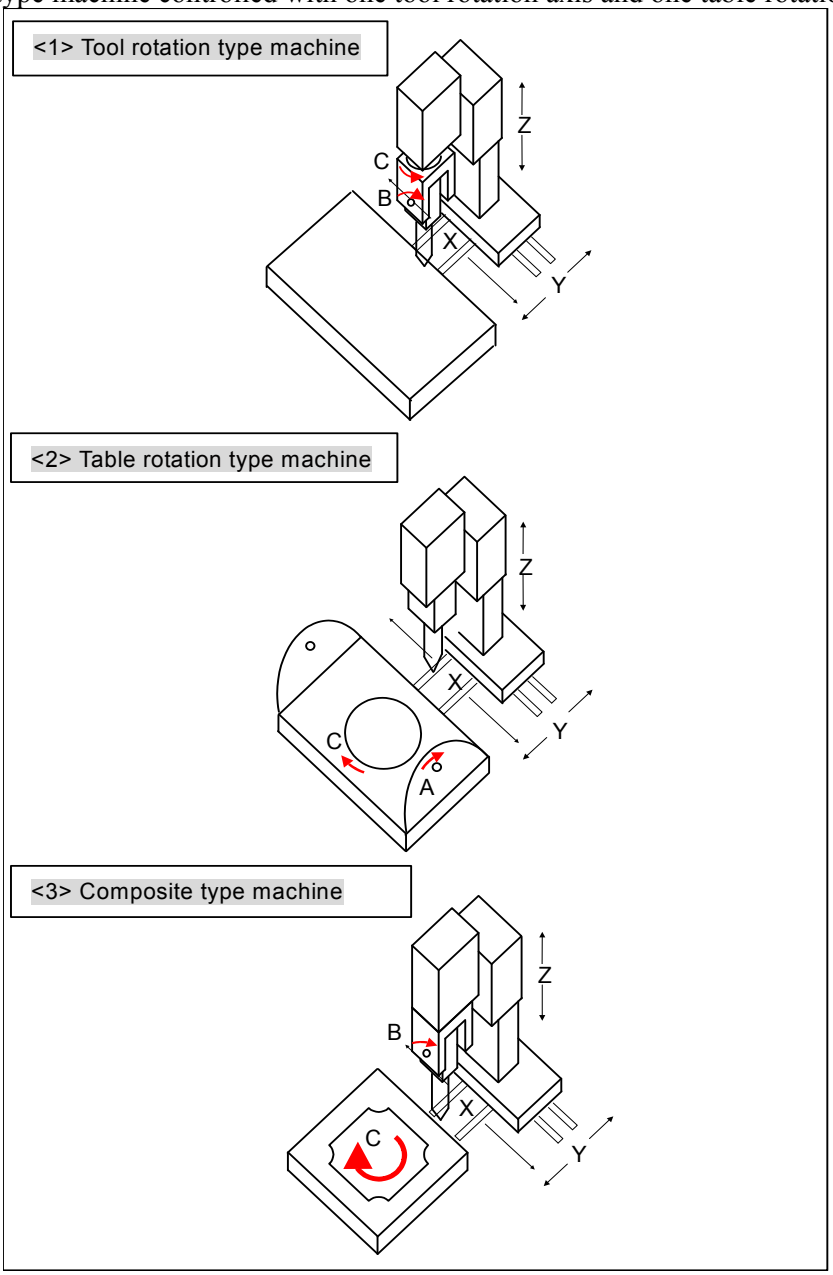


Fig. 22.8 (a) Three types of machine configuration

The coordinate system in which to execute a program for 3-dimensional cutter compensation is called a programming coordinate system.
If, in a 5-axis machine having a table rotation axis, 3-dimensional cutter compensation (tool side offset) is to be performed, the table coordinate system (coordinate system fixed to the table) can be selected as a programming coordinate system.
Alternatively, the workpiece coordinate system fixed to the machine coordinate system can be selected as a programming coordinate system.
There are two types of 3-dimensional cutter compensation (tool side offset), which differ in the method of specifying the tool axis direction.

(1) Type 1
Specify the block end position (for example, A, B, C) of the rotation axis. The CNC performs cutter compensation on the plane perpendicular to the tool axis direction calculated from the specified position on the rotation axis.

(2) Type 2
Instead of specifying a position on the rotation axis, specify the tool axis direction (I, J, K) at the block end point as viewed from the table coordinate system. The CNC calculates the end position on the rotation axis so that the tool points to the specified direction at the end point, and performs cutter compensation on the plane perpendicular to the tool axis direction calculated from the position on the rotation axis.

By using type 2, it is possible to perform the same machining with the same program regardless of the machine configuration of the 5-axis machine (tool rotation type, table rotation type, or composite type).

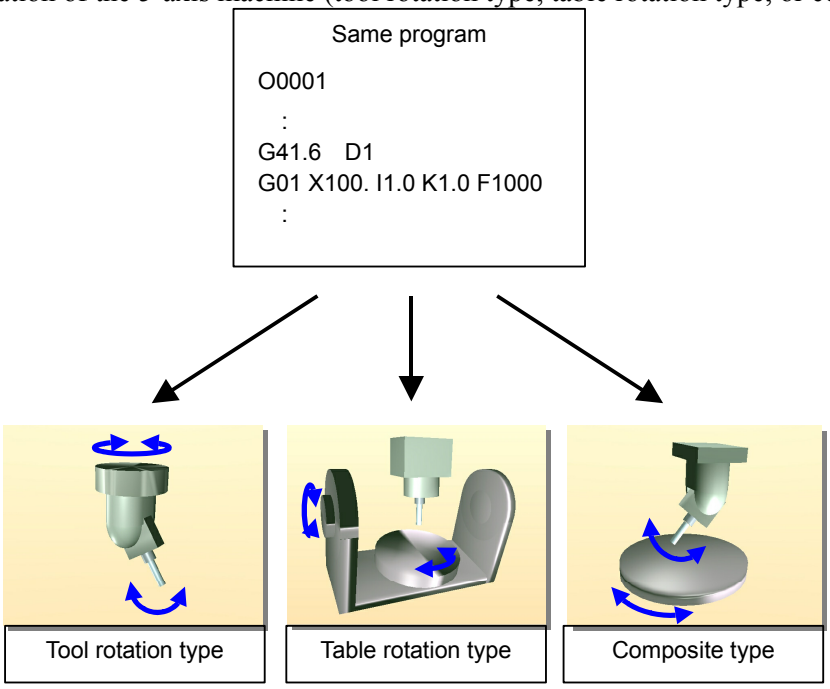


Fig. 22.8 (b) Type 2 Program

22.8.1 Cutter Compensation in Tool Rotation Type Machine

Overview

In a 5-axis machine having two tool rotation axes as shown in Fig. 22.8.1 (a), this function can perform cutter compensation.
Shown below is a 5-axis machine that has tool rotation axis B on the Y-axis and tool rotation axis C on the Z-axis.
This machine configuration is used as a sample configuration in the following explanation unless otherwise noted.

Cutter compensation in tool rotation machines is classified into two types according to the way of machining: tool side offset and leading edge offset.

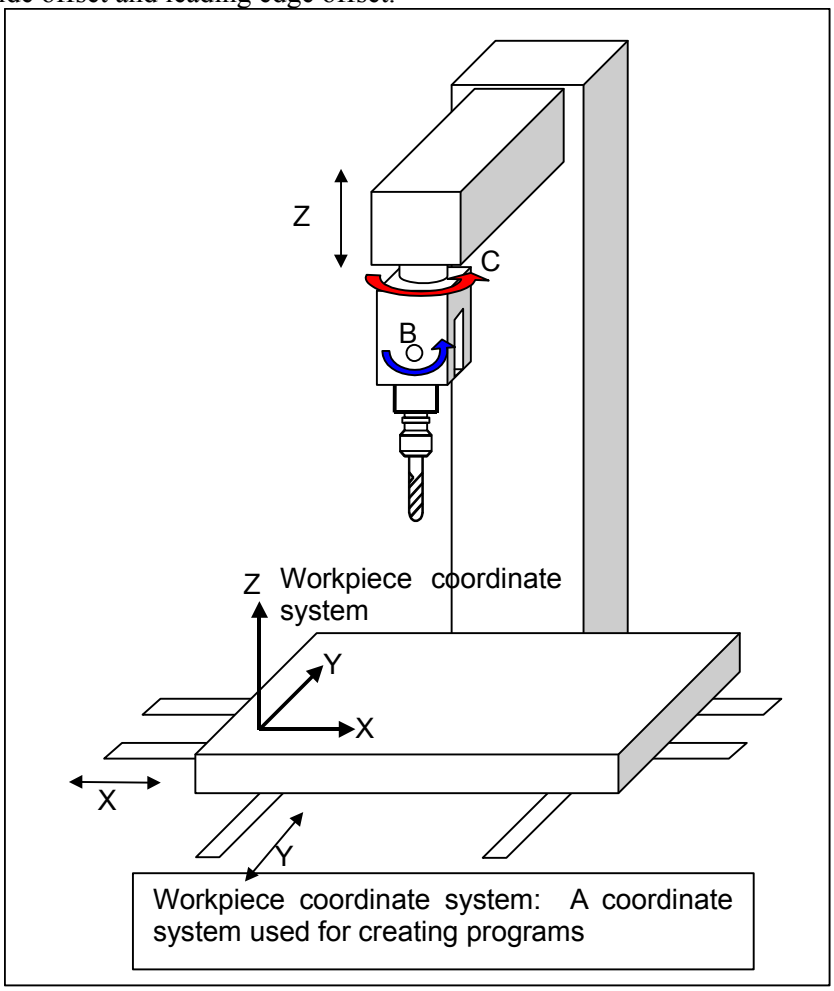


Fig. 22.8.1 (a) Machine having two tool rotation axes

22.8.1.1 Tool side offset

Overview

This type of cutter compensation performs 3-dimensional compensation in a plane (compensation plane) perpendicular to the tool vector.

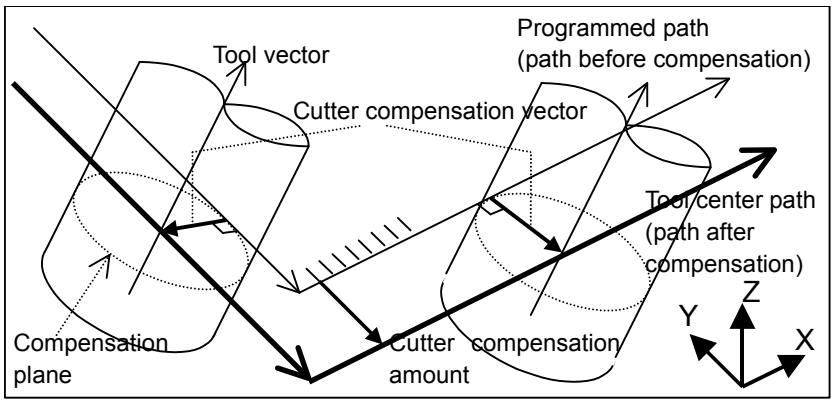


Fig. 22.8.1.1 (a) Tool side offset

Format

- Tool side offset (type 1)

```
G41.2 (or G42.2) IP_ D_ ;
IP_ ;
   :
```

G41.2: Cutter compensation left (group 07)
G42.2: Cutter compensation right (group 07)
IP_: Value specified for axis movement (including rotary axis)
D_: Code specifying the cutter compensation amount (1 to 3 digits)

- Tool side offset (type 2)

```
G41.6 (or G42.6) IP_ I_ J_ K_ D_ Q_ ;
IP_ I_ J_ K_ ;
   :
```

G41.6: Cutter compensation left (group 07)
G42.6: Cutter compensation right (group 07)
IP_: Value specified for axis movement (without rotary axis)
D_: Code specifying the cutter compensation amount (1 to 3 digits)
Q_: Tool's angle of gradient (in degrees)
I_ J_ K_: Tool axis direction at the block end point as viewed from the programming coordinate system

For type 2, do not specify a rotation axis but specify the direction at the tool end point as viewed from the programming coordinate system (workpiece coordinate system), with I, J, and K. Specifying a rotation axis causes alarm PS5460 to be generated.

NOTE
When a movement perpendicular to the next movement (bit 1 (SUV) of parameter No. 5003 is set to 1) is specified as the operation performed at the time of startup or cancellation, a move command such as X_ Y_ Z_ must not be specified in the G41.2 and G42.2 block.

The following are the notes on type 2.

NOTE
1 If one or two of I, J, and K are omitted, the omitted ones of I, J, and K are assumed to be 0.
2 In a block in which all of I, J, and K are omitted, the values of I, J, and K in the previous block are used.
3 If there is only one rotation axis (a hypothetical axis is used), type 2 cannot be used. In this case, if an attempt is made to issue G41.6/G42.6, alarm PS5460 is generated.
4 If using the rotation axis rollover function or rotary axis control function, specify 360 degrees in parameter No. 1260 (mount of travel per rotation about the rotation axis).
5 Only tool side offset provides type 2 commands. Leading edge offset, described later, does not provide type 2 commands.

- Canceling the tool side offset

G40 IP_ ;
G40: Cancellation of cutter compensation (group 07)
IP_: Value specified for axis movement

Explanation

- Tool's angle of gradient in type 2

For type 2 of 3-dimensional cutter compensation, the tool's angle of gradient can be specified with address Q in a G41.6/G42.6 command block. The tool's angle of gradient refers to the angle by which the tool direction to be assumed when machining is actually performed is inclined from the direction specified with (I, J, K) toward the traveling direction on the plane formed by the tool direction specified with (I, J, K) and the traveling direction in the programming coordinate system. (See Fig. 22.8.1.1 (b).)
Because in general, the normal direction of the machining surface is specified with (I, J, K), if it is desired to incline the tool direction to be assumed when machining is actually performed from the normal direction toward the traveling direction, correction may be performed with a Q command.
If the direction specified with (I, J, K) matches the tool direction to be assumed when machining is actually performed, no Q command is necessary.

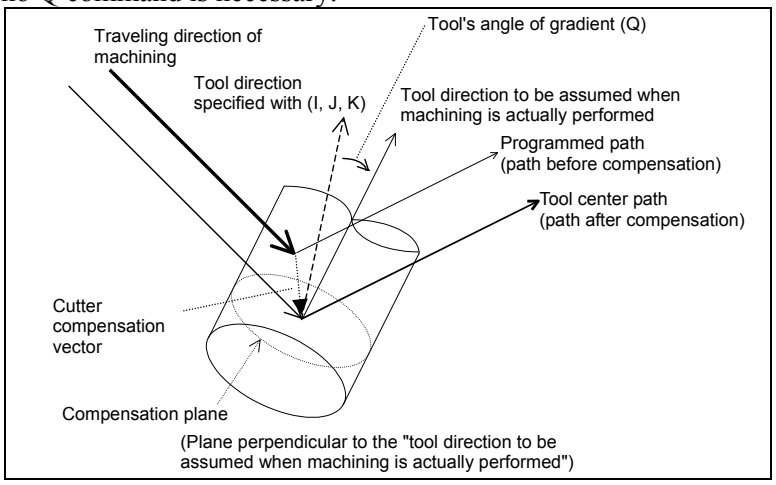


Fig. 22.8.1.1 (b) Tool's angle of gradient in type 2

(Example)
To perform machining by inclining the tool's traveling direction twice, issue a command such as the following:
G41.6 I_ J_ K_ H_ Q2.0

- Operation at startup and cancellation

<1> Type A

The tool is moved in the same way as for cutter compensation as shown below.

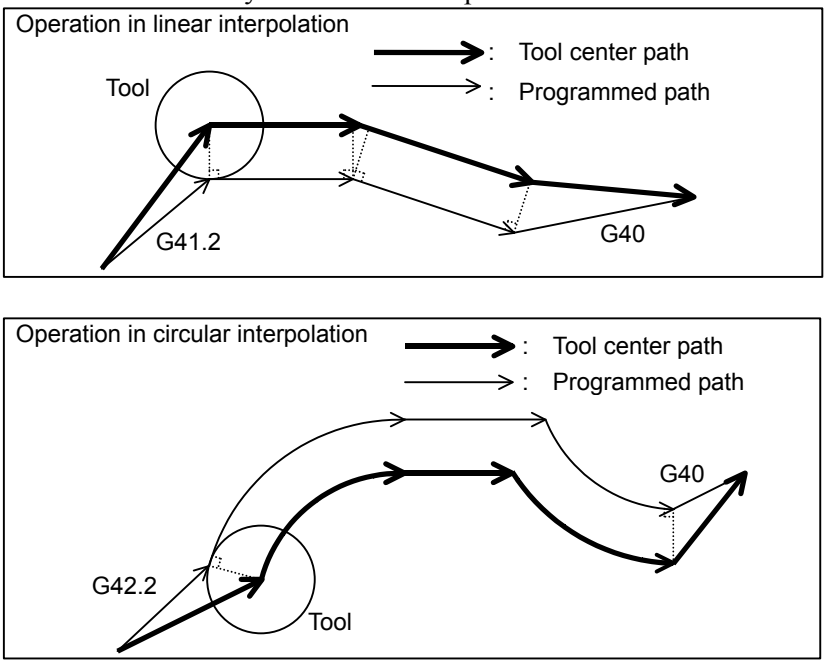


Fig. 22.8.1.1 (c) Operation at startup and cancellation (type A)

When the G40 block contains no movement command, type C cancellation is performed (see <3> Type C below).

<2> Type B

The tool is moved in the same way as for cutter compensation as shown below.

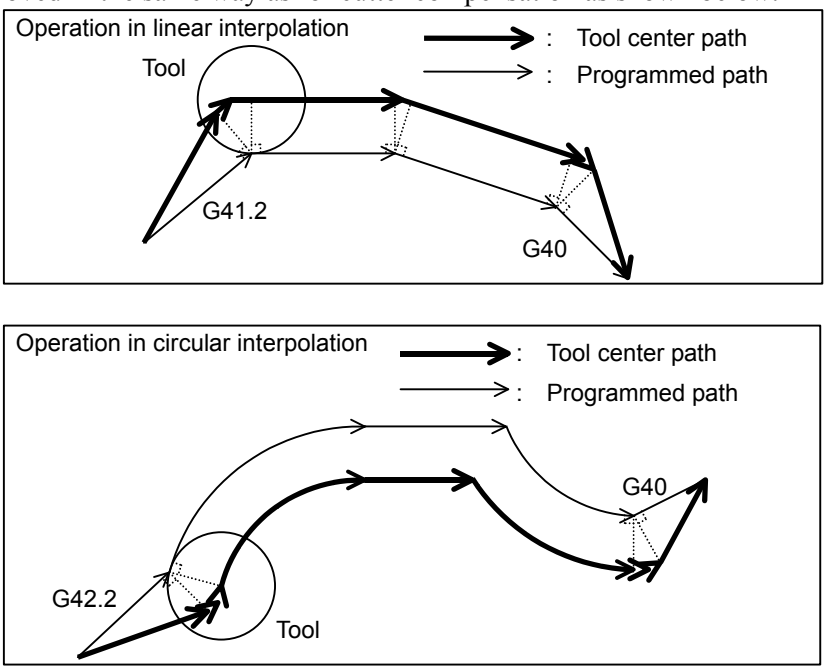


Fig. 22.8.1.1 (d) Operation at startup and cancellation (type B)

When the G40 block contains no movement command, type C cancellation is performed (see <3> Type C below).

<3> Type C
When G41.2, G42.2, or G40 is specified as shown below, a linear block specifying movement by the amount of cutter compensation in the direction orthogonal to the movement direction of the block following startup or the block preceding cancellation is inserted.

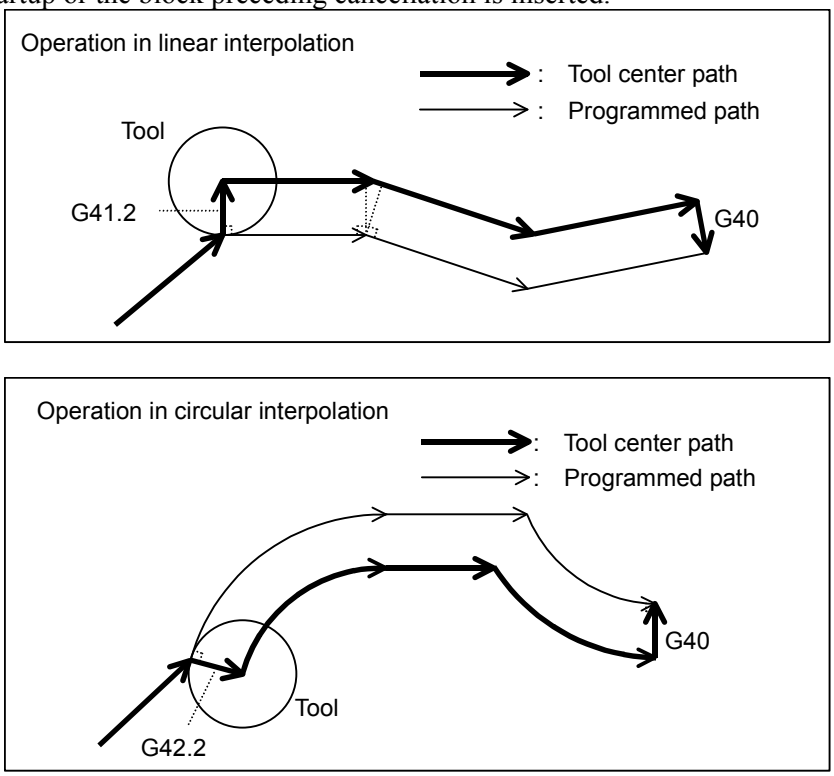


Fig. 22.8.1.1 (e) Operation at startup and cancellation (type C)

NOTE
For type C (bit 1 (SUV) of parameter No. 5003 is 1), the following conditions need to be satisfied during startup and cancellation.
1 A block specifying G40, G41.2, or G42.2 must be in the G00 or G01 mode.
2 A block specifying G40, G41.2, or G42.2 must contain no move command.
3 The block next to a block specifying G41.2 or G42.2 must contain move command G00, G01, G02, or G03.

- Operation during compensation

Operations such as change of the offset direction and offset value, retention of a vector, and interference checks are performed in the same way as for cutter compensation. However, G39 (corner rounding) cannot be specified. So, note the following:

<1> When the tool center path goes outside the programmed path at a corner, a linear movement takes place at the corner without inserting an arc. When the tool center path goes inside the programmed path, nothing is inserted.

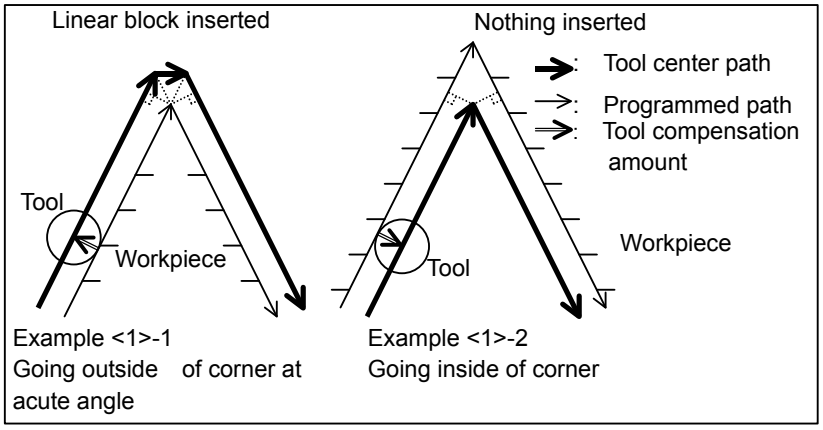


Fig. 22.8.1.1 (f) Operation during compensation <1>-1 and <1>-2

In the above examples, the terms "inside" and "outside" denote how the tool center path is positioned with respect to the programmed path. In the figure below, example <1>-3 shows the same relationship between the tool center path and programmed path as example <1>-1 and indicates that the tool center path is outside the programmed path; example <1>-4 shows the same relationship as example <1>-2 and indicates that the tool center path is inside the programmed path.

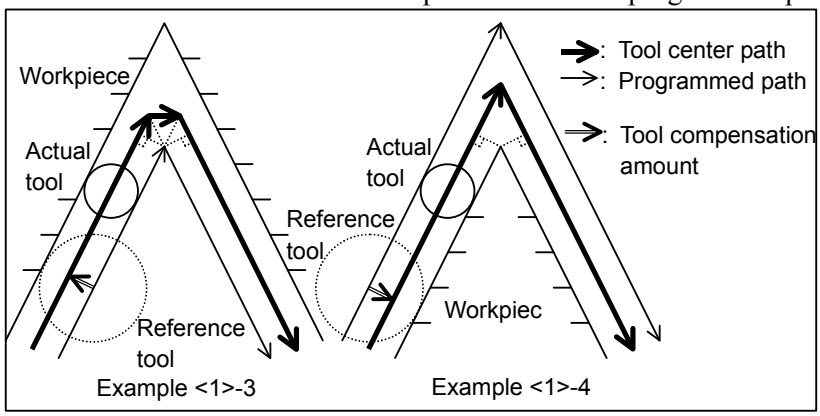


Fig. 22.8.1.1 (g) Operation during compensation <1>-3 and <1>-4

<2> When the tool moves at a corner, the feedrate of the previous block is used if the corner is positioned before a single-block stop point; if the corner is after a single-block stop point, the feedrate of the next block is used.

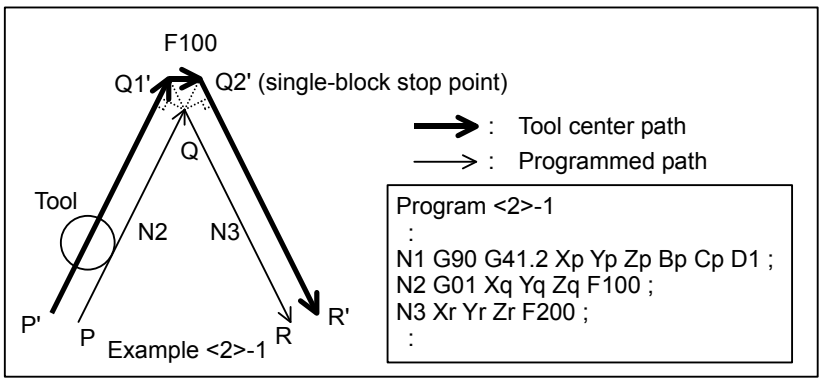


Fig. 22.8.1.1 (h) Operation during compensation <2>

In the above example, the single-block stop point of N2 is Q2', so the feedrates along paths P'-Q1' and Q1'-Q2' are both F100.

<3> When a command that makes the tool retrace the path of the previous block is specified, the tool path can match the locus of the previous block by changing the G code to change the offset direction. If the G code is left unchanged, the operation shown in example <3>-2 results (Fig. 22.8.1.1 (i)):

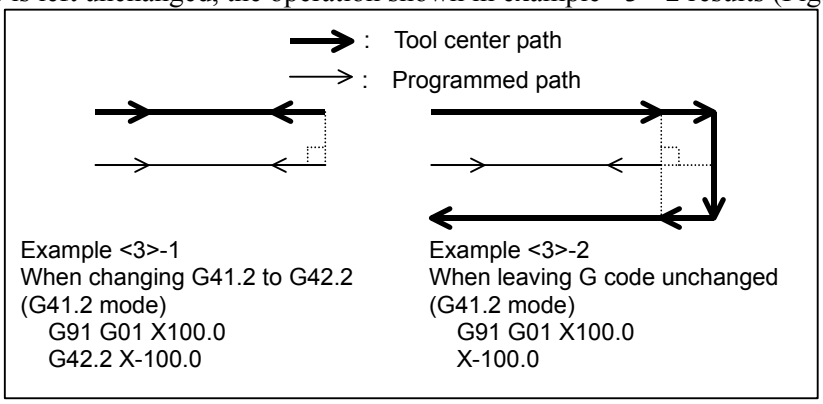


Fig. 22.8.1.1 (i) Operation during compensation <3>

- Interference check when the compensation plane changes

An interference check is made when the compensation plane (a plane perpendicular to the tool vector) has changed.

Example:

If the following program is executed, an alarm PS0041 (INTERFERENCE IN CUTTER COMPENSATION) is issued at N4:

```
O100 F3000
N1 G90 G00 X0 Y0 Z0 A-46 C180
N2 G41.2 D1
N3 G01 X100
N4 Y-200 Z-200
N5 A45
N6 Y-400 Z0
N7 X0
N8 Y-200 Z-200
N9 A-46
N10 Y0 Z0
N11 G40
M30
```

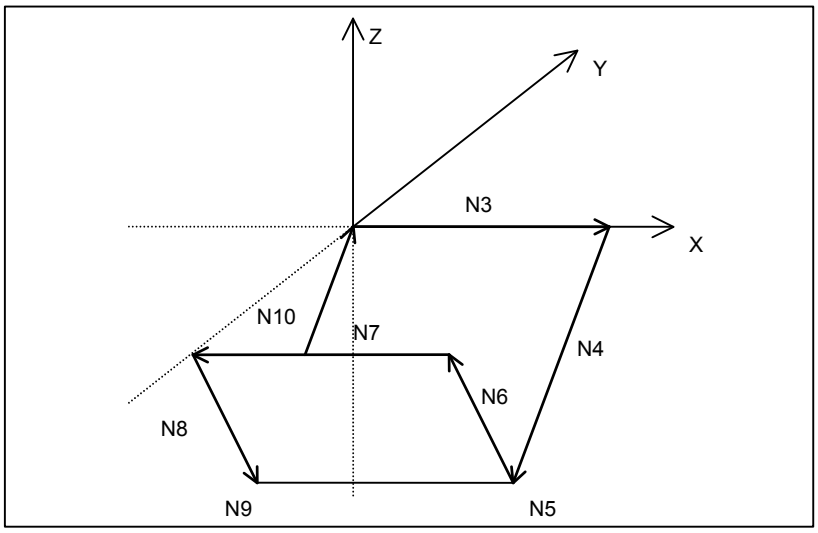


Fig. 22.8.1.1 (j) Conceptual diagram

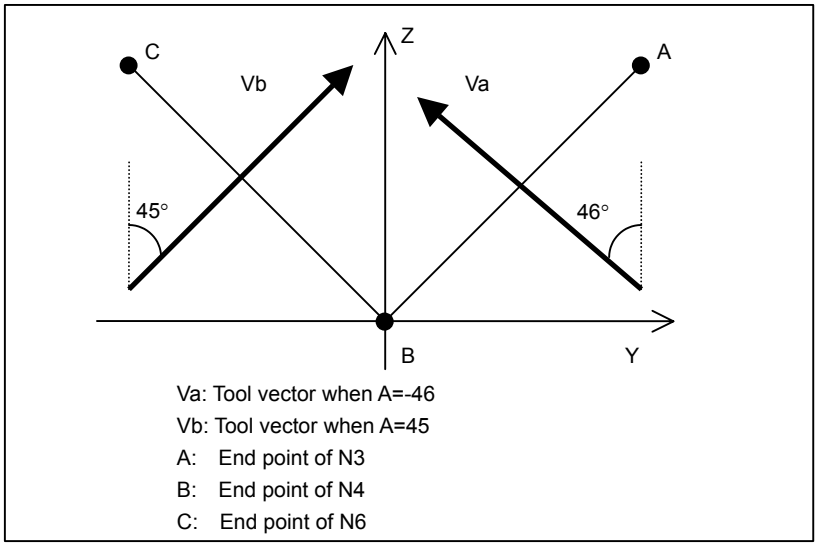


Fig. 22.8.1.1 (k) Tool vector

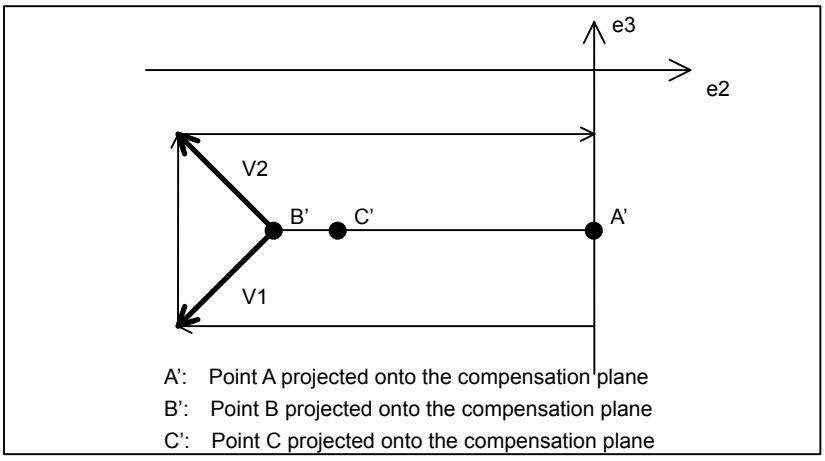


Fig. 22.8.1.1 (l) Compensation vector at the end point (point B) of N4 (in the compensation plane)

The movement direction of A'B' is opposite to that of B'C', so two compensation vectors V1 and V2 are produced at point B' (the end point of N4). In such a case, there is a possibility of overcutting, so an alarm PS0041 is issued at N4.

<1> Conditions for issuing the interference alarm

Suppose that a move command for a rotary axis makes the tool vector change significantly from one block to another. In this case, an interference alarm is assumed because compensation vectors are regarded as being generated in wrong directions when the path angle difference in the compensation plane is large, even though the angle difference between the directions of compensation vectors to be generated by those blocks is small.

Here, the compensation plane is perpendicular to the tool direction (Va in the figure below) of the first of the two blocks.

Specifically, the conditions listed below are used for issuing the alarm.

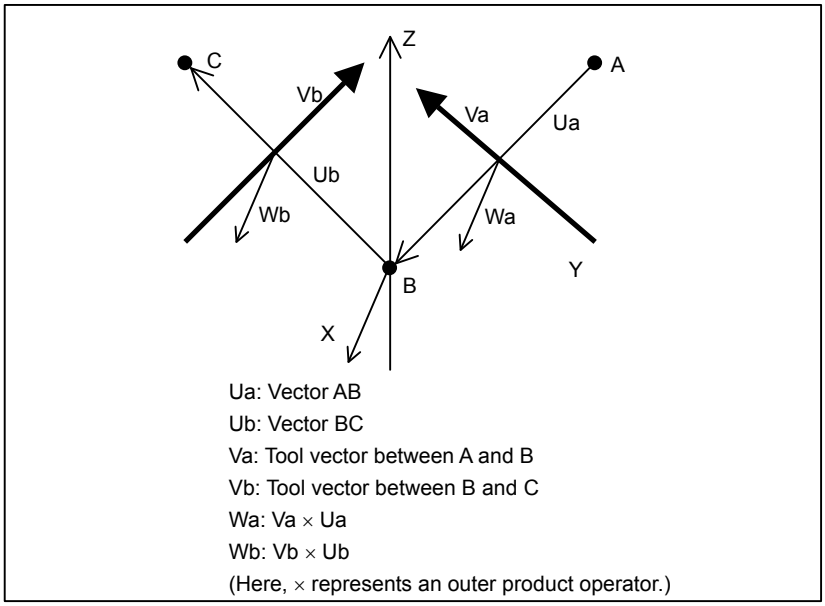


Fig. 22.8.1.1 (m) Conceptual diagram

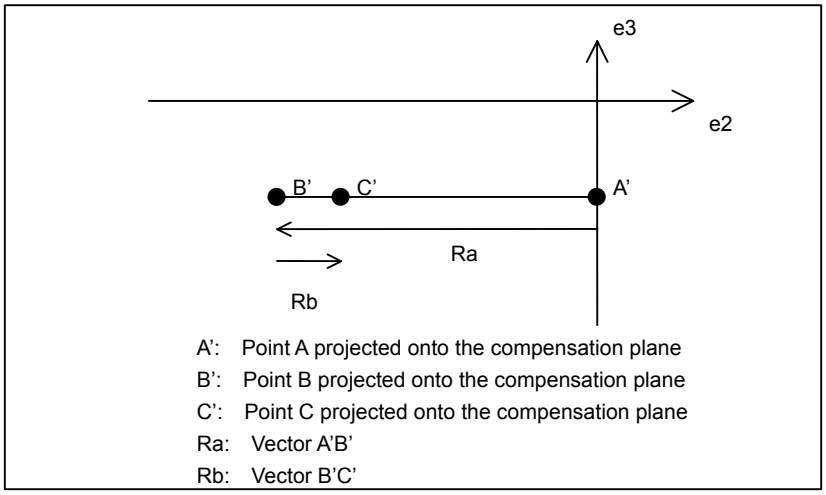


Fig. 22.8.1.1 (n) Programmed path before and after the end point (point B) of N4 (in the compensation plane)

When all the following conditions are satisfied, an alarm PS0041) is issued:

(1) The tool vector changes significantly.
α: Angle for determination set in parameter No. 19635 (The default is 45°.)
$(Va,Vb) \leq \cos(\alpha)$(where, (Va,Vb) means an inner product.)

(2) The difference between the directions of the compensation vectors to be generated is small.
Wa: Direction of a compensation vector to be generated by block AB
Wb: Direction of a compensation vector to be generated by block BC
$Wa = Va \times Ua$
$Wb = Vb \times Ub$
$(Wa,Wb) \geq 0$

(3) The path angle difference in the compensation plane is large.
$(Ra,Rb) < 0$

<2> Suppressing the issue of the alarm with a Q command
By inserting a Q command into a block that resulted in the alarm, the issue of the alarm can be suppressed.

(1) Q1 command
By inserting a Q1 command, a perpendicular vector is generated.

Example: N4 Y-200 Z-200 Q1

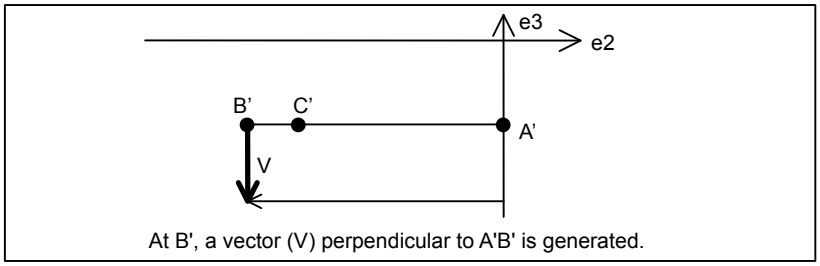


Fig. 22.8.1.1 (o) Q1 command

A perpendicular vector can also be generated by specifying G41.2 or G42.1 in the next block as follows:

Example: N6 G41.2 Y-400 Z0

(2) Q2 command

With a program specifying a linear-to-linear connection, up to two compensation vectors are generated. In this case, the second vector is deleted by inserting a Q2 command.

The Q2 command has no effect on circular interpolation.

Example: N4 Y-200 Z-200 Q2

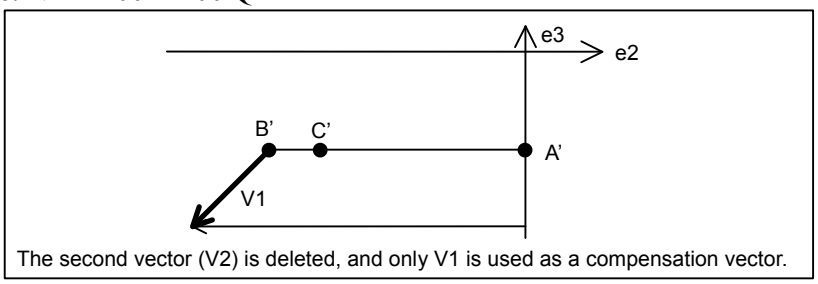


Fig. 22.8.1.1 (p) Q2 command

(3) Q3 command

By inserting a Q3 command, the issue of the alarm can be suppressed.

Example: N4 Y-200 Z-200 Q3

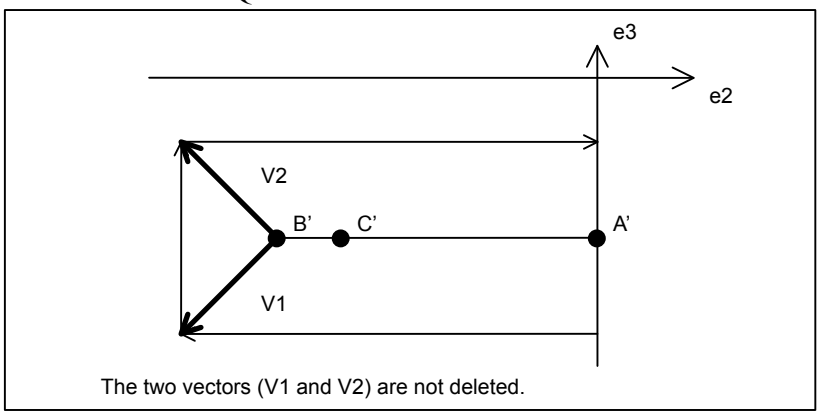


Fig. 22.8.1.1 (q) Q3 command

- Others

When the tool movement changes linear to circular (helical), circular (helical) to linear, or circular (helical) to circular (helical), the start, end, and center points of a circular (helical) movement are projected on the compensation plane that is perpendicular to the tool axis, and a compensation vector is calculated in the plane. The obtained vector is added to the originally specified position to create a position to be specified. Then, the tool is moved linearly or circularly (helically) to the created position.

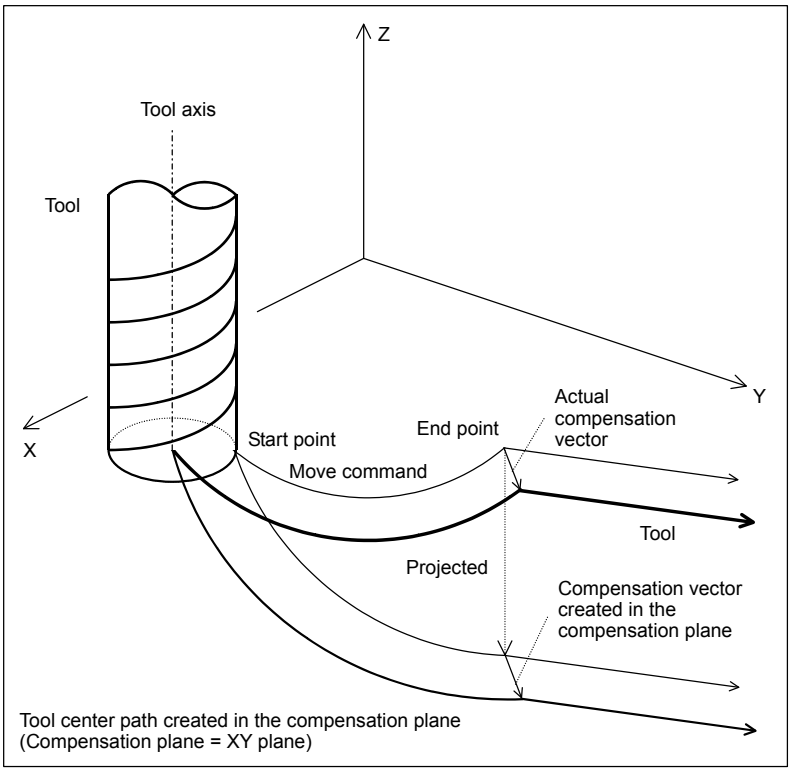


Fig. 22.8.1.1 (r) Operation during compensation

- Angle of the rotary axis for type 2 (when the movement range is not specified)

When the direction of the tool is specified by I, J, K, Q for type 2, more than two pairs of "computed angles" of the rotary axes usually exist.
The "computed angle" is the candidate angle at which the rotary axis is to be controlled in the specified tool axis direction.
The "output angle" is determined from the "computed angle" based on the "output judgment conditions" described below.
The following descriptions assume that there is no movement range specification (parameter No.19741 - No.19744 = 0).

Tool rotation type or table rotation type machine
<1> The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) moving angle is smaller. ↓ ↓ When the master axis moving angle is the same ↓
<2> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) moving angle is smaller. ↓ ↓ When the slave axis moving angle is the same ↓
<3> The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees). ↓ ↓ When the master axis angle is equally near to 0 degree ↓
<4> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

Composite type machine	
<1>	The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) moving angle is smaller. ↓ ↓ When the table moving angle is the same ↓
<2>	The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) moving angle is smaller. ↓ ↓ When the tool moving angle is the same ↓
<3>	The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees). ↓ ↓ When the table angle is equally near to 0 degree ↓
<4>	The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

Fig. 22.8.1.1 (s) Output judgment conditions

The process of judging whether the moving angle is smaller or larger as the output judgement condition is called "movement judgement."
When bit 5 (PRI) of parameter No.19608 is 1, the movement judgements for the first rotary axis and second rotary axis are made in reverse order.

The "movement judgement" process is explained below.
When the "computed angle" is within the range between 0 and 360 degrees, it is called the "basic computed angle."
Usually, two pairs of "basic computed angles" exist.
For example, assume that a tool rotation type or table rotation type machine has rotary axis A (master) and rotary axis B (slave) and that there are two pairs of basic computed angles as follows:
(A θ1 degree; B ϕ1 degree)
(A θ2 degrees; B ϕ2 degrees) where $\theta 1 \leq \theta 2$.
The "computed angle" is obtained from either of the following expressions: "basic computed angle" + 360 degrees × N or "basic computed angle" - 360 degrees × N.
The current position of rotary axis A (master) is PA, and that of rotary axis B (slave) is 0 degree.
Based on the PA angle, the "movement judgement" process is done as follows (when bit 5 (PRI) of parameter No.19608 is 0).

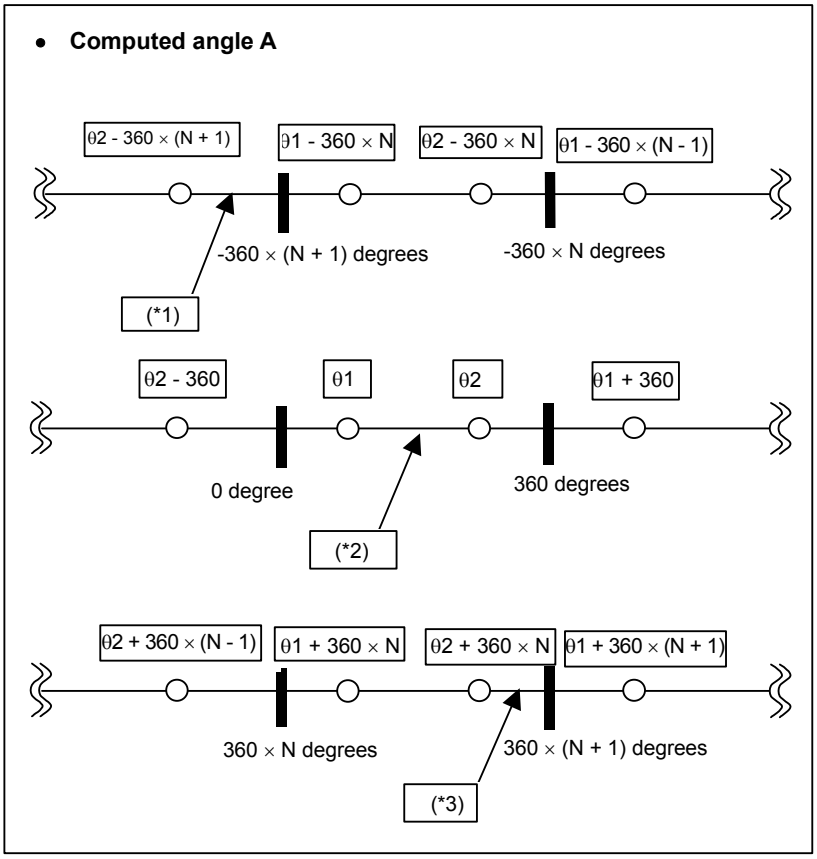


Fig. 22.8.1.1 (t) Movement judgment

When the PA angle is (*1):
The output angle is: (A θ2 - 360 × (N + 1) degrees; B ϕ2 degrees).
Namely, θ2 - 360 × (N + 1) degrees is adopted that is nearer to the computed angle of A, and ϕ2, which is the same group as θ2, is adopted as the computed angle of B.

When the PA angle is (*2):
The output angle is: (A θ1 degrees; B ϕ1 degrees).
Namely, θ1 degrees is adopted that is nearer to the computed angle of A, and ϕ1, which is the same group as θ1, is adopted as the computed angle of B.

When the PA angle is (*3):
The output angle is: (A θ2 + 360 × N degrees; B ϕ2 degrees).
Namely, θ2 + 360 × N degrees is adopted that is nearer to the computed angle of A, and ϕ2, which is the same group as θ2, is adopted as the computed angle of B.

When the moving angle of rotary axis A (master) is the same, a "movement judgement" is made for rotary axis B (slave) according to the "output judgment conditions."
If the "output angle" of rotary axis A is determined by the "movement judgement" for rotary axis A, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis B.
Similarly, if the "output angle" of rotary axis B is determined by the "movement judgement" for rotary axis B, the computed angle representing the "smaller moving angle" is adopted as the "output angle" of rotary axis A.

The "output angle" is explained below using a tool rotation type machine as an example.
This example illustrates a machine having a "BC type tool axis Z."

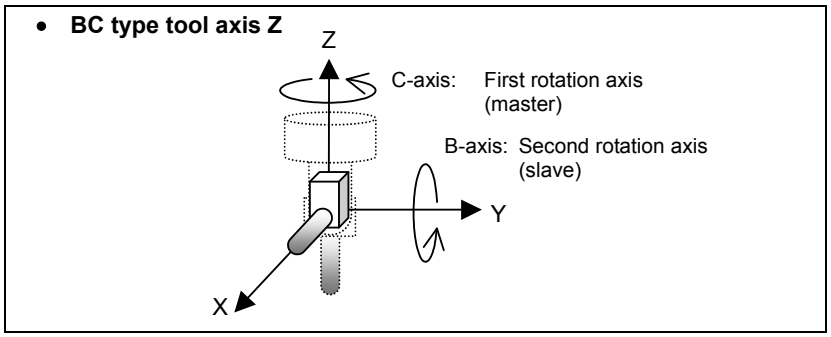


Fig. 22.8.1.1 (u) BC type tool axis Z

The following two pairs of "computed basic angles" exist that direct the tool axis toward the +X-axis direction.
(B 90 degrees; C 180 degrees)
(B 270 degrees; C 0 degree)

<1> When the current rotary axis angles are (B -70 degrees; C 30 degrees)
The "output angles" are (B -90 degrees; C 0 degree).
0 degree is adopted because it is nearer to the current position (30 degrees) of the C-axis that is the master axis. For the B-axis, 270 degrees is adopted which is the same group. However, this is changed to -90 degrees (270 degrees - 360 degrees) which is the nearest to the current position of the B-axis (-70 degrees).

<2> When the current rotary axis angles are (B 80 degrees; C 500 degrees)
The "output angles" are (B 90 degrees; C 540 degrees).
540 degrees (180 degrees + 360 degrees) is adopted because it is nearer to the current position (500 degrees) of the C-axis that is the master axis. For the B-axis, 90 degrees is adopted which is the same group.

<3> When the current rotary axis angles are (B 60 degrees; C 90 degrees)
The "output angles" are (B 90 degrees; C 180 degrees).
Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. 90 degrees is adopted because it is nearer to the current position (60 degrees) of the B-axis that is the slave axis. For the C-axis, 180 degrees is adopted which is the same group.

<4> When the current rotary axis angles are (B 180 degrees; C 90 degrees)
The "output angles" are (B 270 degrees; C 0 degree).
Since the two candidates are equally near to the current position (90 degrees) of the C-axis that is the master axis, a judgment is made based on the current position of the B-axis. In this case, however, the two candidates are also equally near to the current position of the B-axis (180 degrees). Therefore, the candidate is adopted in which the C-axis (master axis) is nearer to 0 degree.
That is, the pair is adopted whose C-axis angle is 0 degree and whose B-axis angle is 270 degrees.
When the slave axis angle is 0 degree, the direction of the tool axis becomes fixed regardless of the master axis angle.

In that case, the master axis does not move from the current angle.

An explanation is shown below using a machine having a "BC type tool axis Z" as an example.

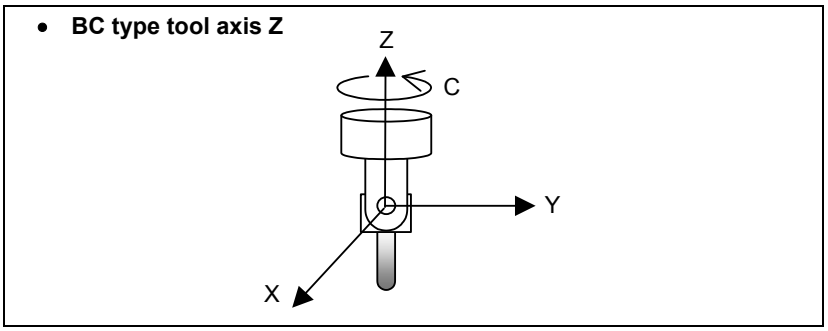


Fig. 22.8.1.1 (v) BC type tool axis Z

When the current rotary axis angles are (B 45 degrees; C 90 degrees), the "output angles" are (B 0 degree; C 90 degrees).

- Angle of the rotary axis for type 2 (when the movement range is specified)

If the upper and lower limits of the movement range of the rotary axis is specified using parameters No.19741 to No.19744, the rotary axis will move only within the specified range when the direction is specified using I, J, K, Q command for type 2 control.

Although the procedure for determining the angles is the same as that used "when the movement range is not specified," the "output angles" need to be selected from **those computed angles that are within the specified movement range for both axes**.

Tool rotation type or table rotation type machine
<1> Of the angle pairs whose master and slave axis angles are both within the specified movement range, the rotary axis angle pair whose master axis (first rotary axis) moving angle is smaller represents the "output angles."
↓
↓ When the master axis moving angle is the same
↓
<2> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) moving angle is smaller.
↓
↓ When the slave axis moving angle is the same
↓
<3> The "output angles" are represented by the computed rotary axis angle pair whose master axis (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).
↓
↓ When the master axis angle is equally near to 0 degree
↓
<4> The "output angles" are represented by the computed rotary axis angle pair whose slave axis (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

Composite type machine	
<1>	Of the angle pairs whose master and slave axis angles are both within the specified movement range, the rotary axis angle pair whose table (second rotary axis) moving angle is smaller represents the "output angles."
	↓
	↓ When the table moving angle is the same
	↓
<2>	The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) moving angle is smaller.
	↓
	↓ When the tool moving angle is the same
	↓
<3>	The "output angles" are represented by the computed rotary axis angle pair whose table (second rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).
	↓
	↓ When the table angle is equally near to 0 degree
	↓
<4>	The "output angles" are represented by the computed rotary axis angle pair whose tool (first rotary axis) angle is nearer to 0 degree (multiple of 360 degrees).

Fig. 22.8.1.1 (w) Output judgment conditions

When bit 5 (PRI) of parameter No.19608 is 1, the movement judgements for the first rotary axis and second rotary axis are made in reverse order.

⚠ CAUTION

1 If the lower limit of the movement range is larger than the upper limit, alarm PS5459 occurs when G43.5 is specified.
2 If no "computed angle" is found within the movement range because the range is too small, alarm PS5459 occurs.
3 If 0 is set for both parameters that specify the upper and lower limits of the movement range, the tool operates assuming that there is no range specification.
4 When the rotary axis rollover function or rotary axis control function is used (in which case, set parameter No.1260 (amount of movement per rotation of the rotary axis) to 360 degrees), the tool does not move beyond 0 degree (360 degrees) (does not take the shortcut) if the movement range is set between 0 and 360 degrees. Also, do not specify a negative value or a value larger than 360 degrees for the movement range.

An example of the "movement judgement" process is given below.
Assume that a tool rotation type or table rotation type machine has rotary axis A (master) and rotary axis B (slave) and that there are two pairs of basic computed angles as follows:
(A θ1 degree; B ϕ1 degree)
(A θ2 degrees; B ϕ2 degrees) where $\theta 1 \leq \theta 2$.
The "computed angle" is obtained from either of the following expressions: "basic computed angle" + 360 degrees × N or "basic computed angle" - 360 degrees × N.
Assume that the current positions and movement ranges of rotary axis A (master) and rotary axis B (slave) are as shown in Fig. 22.8.1.1 (x), Fig. 22.8.1.1 (y).

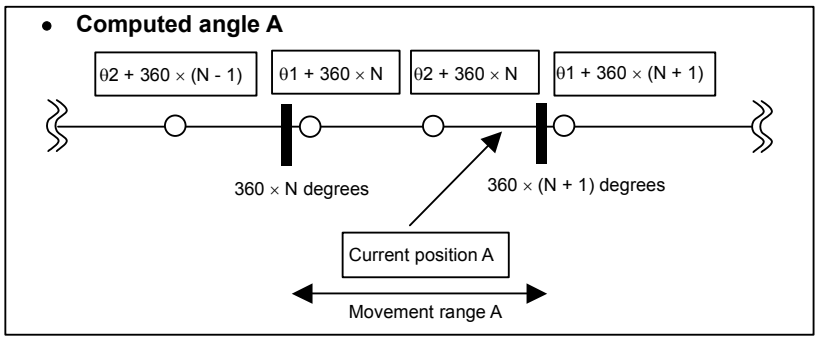


Fig. 22.8.1.1 (x) Computed angle of rotary axis A and its current position and movement range

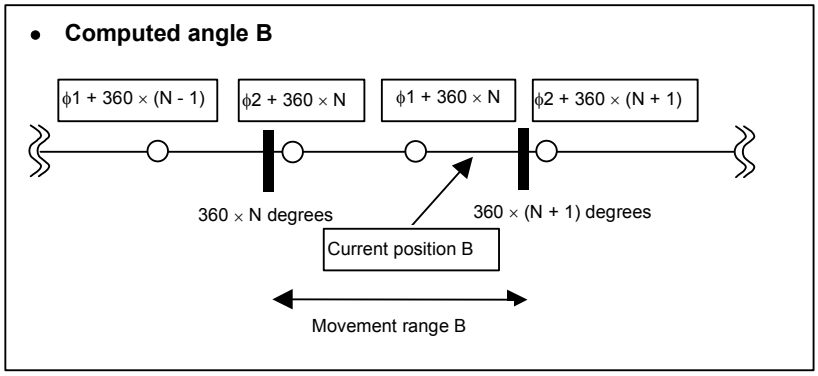


Fig. 22.8.1.1 (y) Computed angle of rotary axis B and its current position and movement range

When the two axes have a positional relationship as shown in the figure, the output angle of rotary axis A is (θ2 + 360 × N) degrees and that of rotary axis B is (ϕ2 + 360 × N) degrees (when bit 5 (PRI) of parameter No.19608 is set to 0).
More concretely, from the computed angles obtained for rotary axis A, the nearest angle within the movement range, i.e. θ2 + 360 × N degrees, is first adopted. Then, from the computed angles obtained for rotary axis B, the angle belonging to the same group as θ2, i.e. ϕ2 + 360 × N, is adopted.

Note that, in this example, the output angles and moving direction differ depending on whether the movement range is specified or not (0 to 360 degrees), even if N is set to 0 and coordinates are rounded to 0 to 360 degrees.
Namely, if the movement range is not specified, θ1 + 360 degrees nearest to the current position is adopted as the computed angle for rotary axis A and, from the computed angles belonging to the same group as θ1, ϕ1 degrees nearest to the current position is adopted as the computed angle for rotary axis B. Rotary axis A moves in the plus direction. As its coordinate is rounded to 360 degrees, rotary axis A reaches θ1 degrees while moving in the plus direction.
By contrast, when the movement range is set to 0 to 360 degrees, the output angles are (A θ2 degrees; B ϕ2 degrees). Neither rotary axis A nor B moves in a way that it exceeds 0 degree (360 degrees).

22.8.1.2 Leading edge offset

Overview

Leading edge offset is a type of cutter compensation used when a workpiece is machined with the edge of a tool. The tool is automatically shifted by the amount of cutter compensation on the line where a plane formed by a tool vector and tool movement direction meets a plane perpendicular to the tool axis direction.

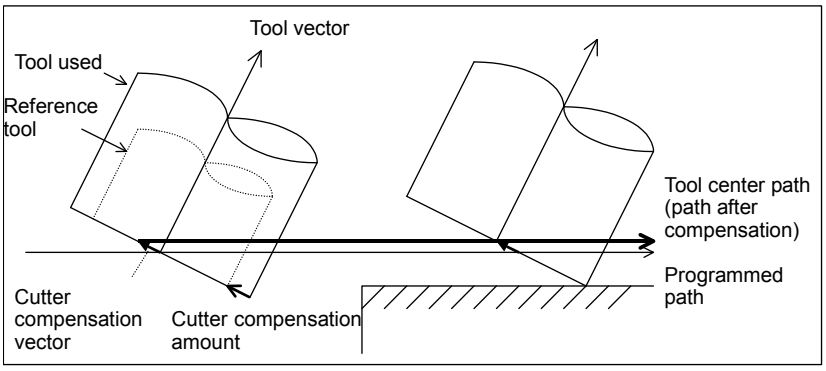


Fig. 22.8.1.2 (a) Leading edge offset

Format

- Leading edge offset

```
G41.3 D_ ;
```

- Canceling the leading edge offset

```
G40 ;
```

NOTE

1 G41.3 can be specified only in the G00 and G01 modes. In a block containing G41.3 or G40, only addresses D, O, and N can be specified.
2 The block that follows a block containing a G41.3 command must contain a move command.
In the block after G41.3, however, a tool movement in the same direction as the tool axis direction or the opposite direction cannot be specified.
3 No modal G code that belongs to the same group as G00 and G01 can be specified in the G41.3 mode. If such a modal G code is specified, alarm PS5460 is issued.
4 Leading edge offset does not provide type 2 commands. It does not allow the tool direction to be specified with the I, J, and K commands.

Explanation

- Operation at startup and cancellation

The operation performed at leading edge offset startup and cancellation does not vary. When G41.3 is specified, the tool is moved by the amount of compensation (Vc) in the plane formed by the movement vector (V_M) of the block after the G41.3 block and the tool vector (V_T) obtained at the time of G41.3 specification. The tool movement is perpendicular to the tool vector. When G40 is specified, the tool is moved to cancel V_C. The following illustrates how the compensation is performed:

<1> When the tool vector is inclined in the tool movement direction

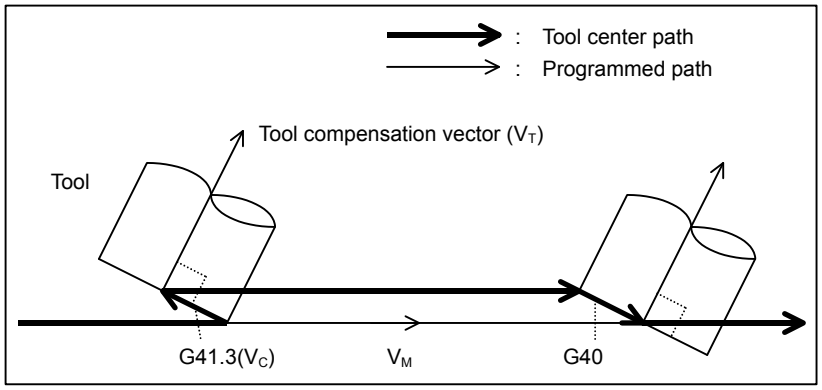


Fig. 22.8.1.2 (b) When the tool vector is inclined in the tool movement direction

<2> When the tool vector is inclined in the direction opposite to the tool movement direction

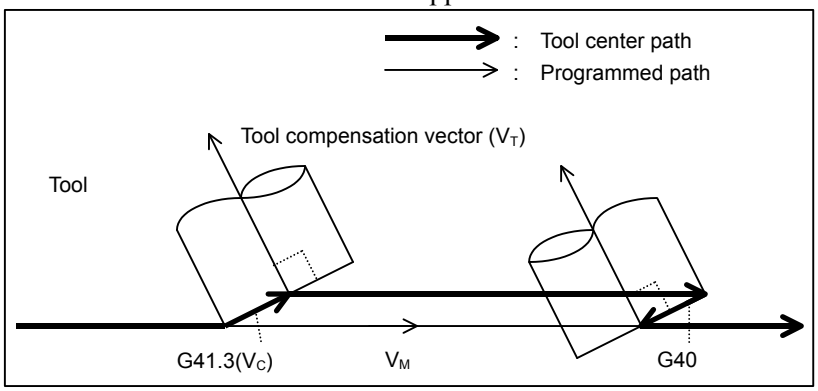


Fig. 22.8.1.2 (c) When the tool vector is inclined in the direction opposite to the tool movement direction

- Operation during compensation

The tool center moves so that a compensation vector (VC) perpendicular to the tool vector (VT) is created in the plane formed by the tool vector (VT) at the end point of each block and the movement vector (VM) of the next block.

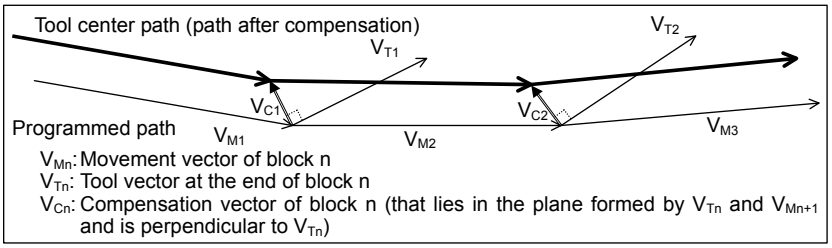


Fig. 22.8.1.2 (d) Operation during compensation

If a G code or M code that suppresses buffering is specified in the compensation mode, however, the compensation vector created immediately before the specification is maintained.

When a block specifying no movement (including a block containing a move command for a rotary axis only) is specified in the compensation mode, the movement vector of the block after the block specifying no movement is used to create a compensation vector as shown below.

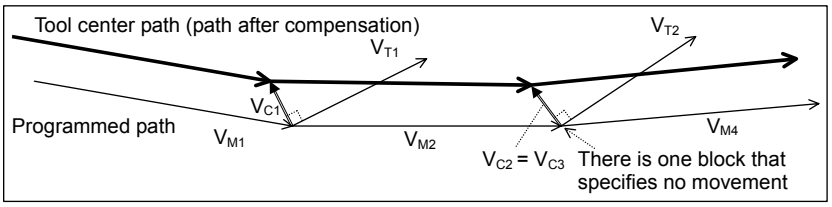


Fig. 22.8.1.2 (e) When there is one block that specifies no movement

If block 3 specifies no movement, the compensation vector of block 2 (VC2) is created in a plane formed by the movement vector of block 4 (VM4) and the tool vector (VT2) at the end of block 2. VC2 is perpendicular to VT2.

> ⚠ **CAUTION**
> If two or more successive blocks specify no movement, the previously created compensation vector is maintained. However, such specification should be avoided.

- Block immediately before the offset cancel command (G40)

In the block immediately before the offset cancel command (G40), a compensation vector is created from the movement vector of that block and the tool vector at the end point of the block as shown below.

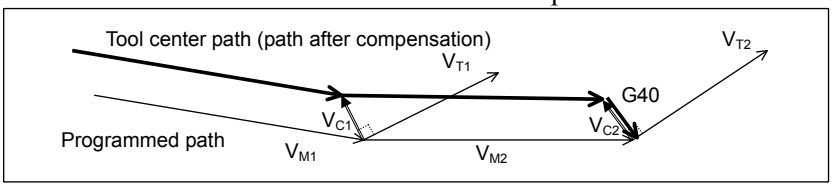


Fig. 22.8.1.2 (f) Block immediately before G40

The compensation vector (VC2) of block 2 is created in a plane formed by the tool vector (VT2) at the end point of block 2 and the movement vector (VM2) of block 2. VC2 is perpendicular to VT2.

- Compensation performed when θ is approximately 0°, 90°, or 180°

When the included angle θ between VMn+1 and VTn is regarded as 0°, 180°, or 90°, the compensation vector is created in a different way. So, when creating a program, note the following points:

<1> Setting a variation range for determining θ to be 0°, 180°, or 90°

When the included angle (θ) between the tool vector (VTn) and movement vector (VMn+1) becomes approximately 0°, 180°, or 90°, the system regards θ as 0°, 180°, or 90°, respectively, then creates a compensation vector which is different from the normal compensation vector. The variation range used for determining θ to be 0°, 180°, and 90° is set in parameter No. 19631. For example, let the angle set in this parameter be Δθ. Then, the system regards θ as follows:

(1) If $0 \leq \theta \leq \Delta\theta$, θ is regarded as 0°.

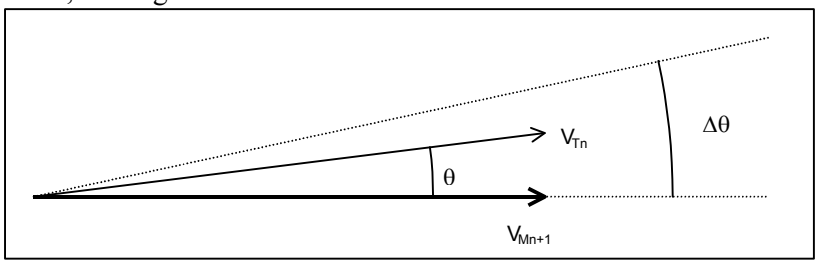


Fig. 22.8.1.2 (g) Determination of θ = 0°

(2) If $(180 - \Delta\theta) \leq \theta \leq 180$, θ is regarded as 180°.

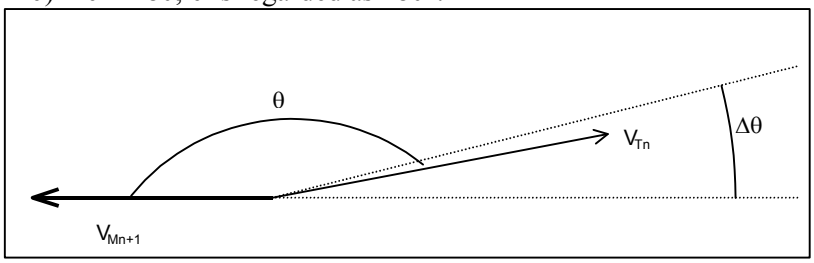


Fig. 22.8.1.2 (h) Determination of θ = 180°

(3) If $(90 - \Delta\theta) \leq \theta \leq (90 + \Delta\theta)$, θ is regarded as 90°.

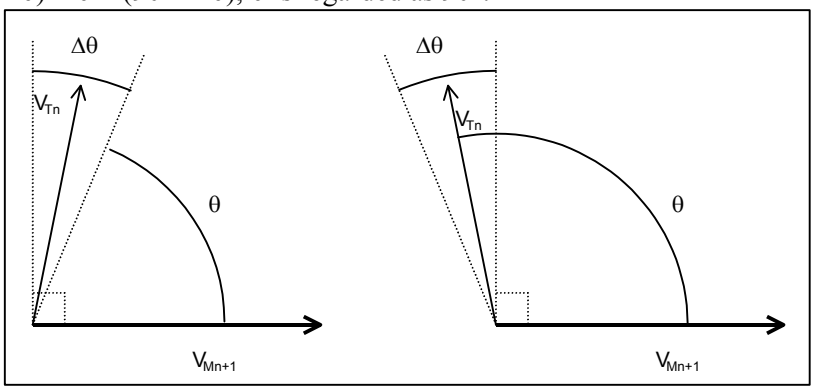


Fig. 22.8.1.2 (i) Determination of θ = 90°

<2> Compensation vector when θ is regarded as 0° or 180°

At startup (when G41.3 is specified), alarm PS5408 is issued.

This means that the tool vector of a block and the movement vector of the next block must not point in the same direction or in opposite directions at startup.

At other than startup, the previously created compensation vector is maintained without change.

If the included angles between VT2 and VM3, VT3 and VM4, and VT4 and VM5 are regarded as 0°, compensation vector VC1 of block 1 is maintained as compensation vectors VC2, VC3, and VC4 of blocks 2, 3, and 4, respectively.

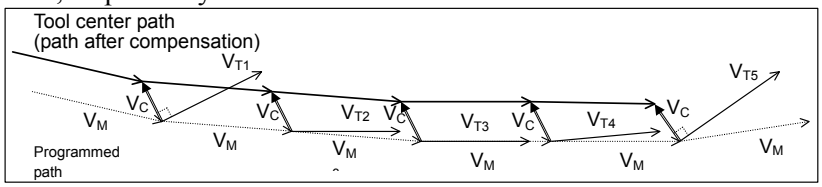


Fig. 22.8.1.2 (j) When θ = 0° is determined

If the included angles between VT2 and VM3, VT3 and VM4, and VT4 and VM5 are regarded as 180°, compensation vector VC1 of block 1 is maintained as compensation vectors VC2, VC3, and VC4 of blocks 2, 3, and 4, respectively.

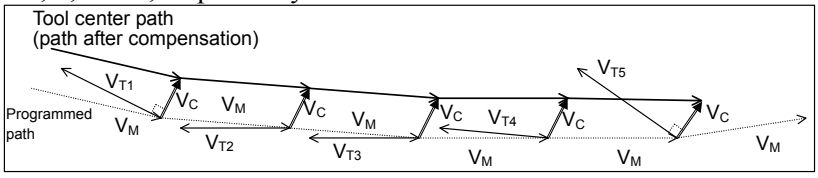


Fig. 22.8.1.2 (k) When θ = 180° is determined

<3> Compensation vector when θ is regarded as 90°

If the previous compensation vector (VCn-1) points in the opposite direction ((VMn × VTn-1) × VTn-1 direction) to VMn with respect to VTn-1, the current compensation vector (VCn) is created so that it also points in the (VMn+1 × VTn) × VTn direction.

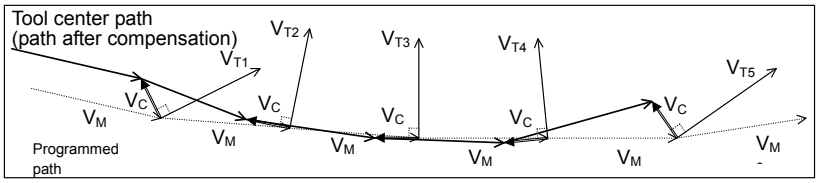


Fig. 22.8.1.2 (l) When θ = 90° is determined 1

If the previous compensation vector (VCn-1) points in the same direction (-(VMn × VTn-1) × VTn-1 direction) as VMn with respect to VTn-1, the current compensation vector (VCn) is created so that it also points in the -(VMn+1 × VTn) × VTn direction.

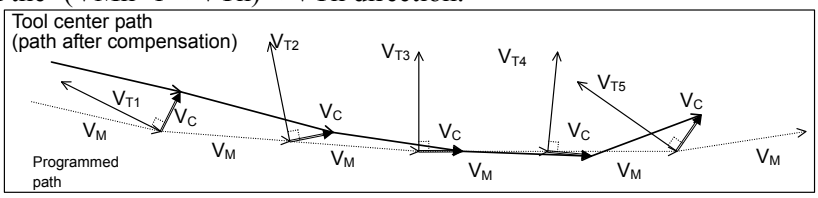


Fig. 22.8.1.2 (m) When θ = 90° is determined 2

22.8.1.3 Tool tip position (cutting point) command

Overview

For machines having a rotary axis for rotating a tool, this function performs 3-dimensional cutter compensation at the tool tip position if a programmed point is specified with a pivot point.
When this function is used, the programmed point (pivot point) is converted into a tool tip position (cutting point) and a vector of 3-dimensional cutter compensation is calculated for the position obtained by the conversion. Then, the programmed point (pivot point) is compensated for with the vector of 3-dimensional cutter compensation.
If the tool side offset (G41.2/G42.2) of 3-dimensional cutter compensation is performed, the operation of this function is as follows:

(1) If parameter No. 19632 is 0
The vector of 3-dimensional cutter compensation is calculated at the programmed point (pivot point).
(2) If parameter No. 19632 is not 0 (this function)
The vector of 3-dimensional cutter compensation is calculated at the tool tip position (cutting point).

Explanation

- Operation explanation

This function calculates a vector at the tool tip position for the 3-dimensional cutter compensation function as described below.

(1) Convert the programmed coordinates from a programmed point (pivot point) to a tool tip position (cutting point). Parameter No. 19632 is used to store the distance from the programmed point (pivot point) to the tool tip position (cutting point).
(2) Calculate a vector of 3-dimensional cutter compensation at the tool tip position (cutting point).
(3) Add the cutter compensation vector to the programmed point (pivot point).

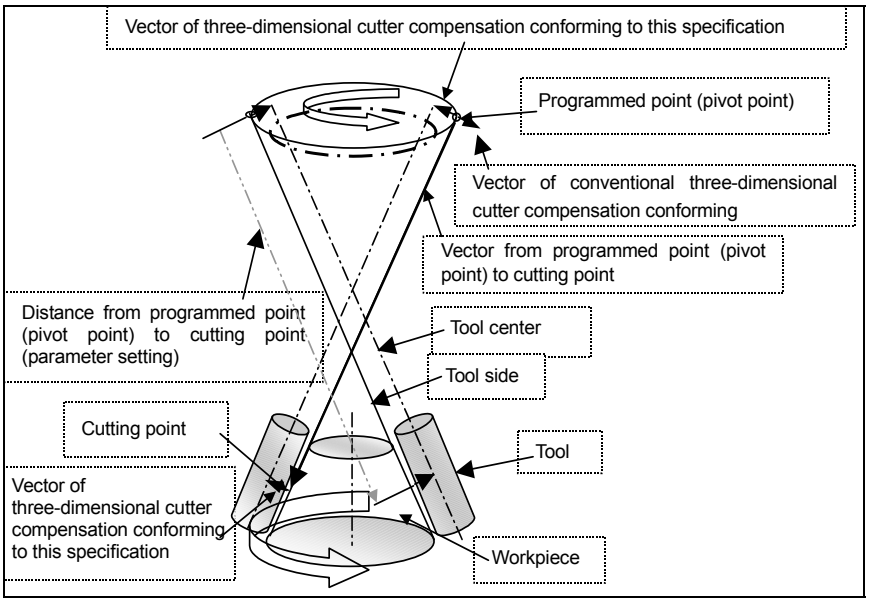


Fig. 22.8.1.3 (a) Basic operation (for G42.2)

- Operation example

For a machine configuration in which the tool axis direction is along the Z-axis and the rotary axes are the B and C axes (Fig. 22.8.1.3 (b))

LC :	Parameter No. 19632 specifying the distance from the programmed point (pivot point) to the tool tip position (cutting point)
B :	Specified B-axis value, c: Specified C-axis value
Q = (Qx,Qy,Qz) :	Programmed point (pivot point)
P, R :	Programmed points (pivot points) in the preceding and succeeding blocks
QT = (QTx,QTy,QTz) :	Tool position (tool tip position (cutting point)) resulting from conversion
PT, RT :	Tool positions (tool tip positions (cutting positions)) in the preceding and succeeding blocks

Then,

<1> Convert programmed points (pivot points) P, Q, and R to tool tip positions (cutting points) PT, QT, and RT.

$QTx = LC \times \sin(b) \times \cos(c) + Qx$

$QTy = LC \times \sin(b) \times \sin(c) + Qy$

$QTz \quad LC \times \cos(b) + Qz$

(The same applies to PT and RT.)

<2> Calculate vector VD of 3-dimensional cutter compensation from tool tip positions (cutting points) PT, QT, and RT and tool gradient VT.

<3> Add cutter vector VD to programmed point (pivot point) Q and set the result as the end point position.

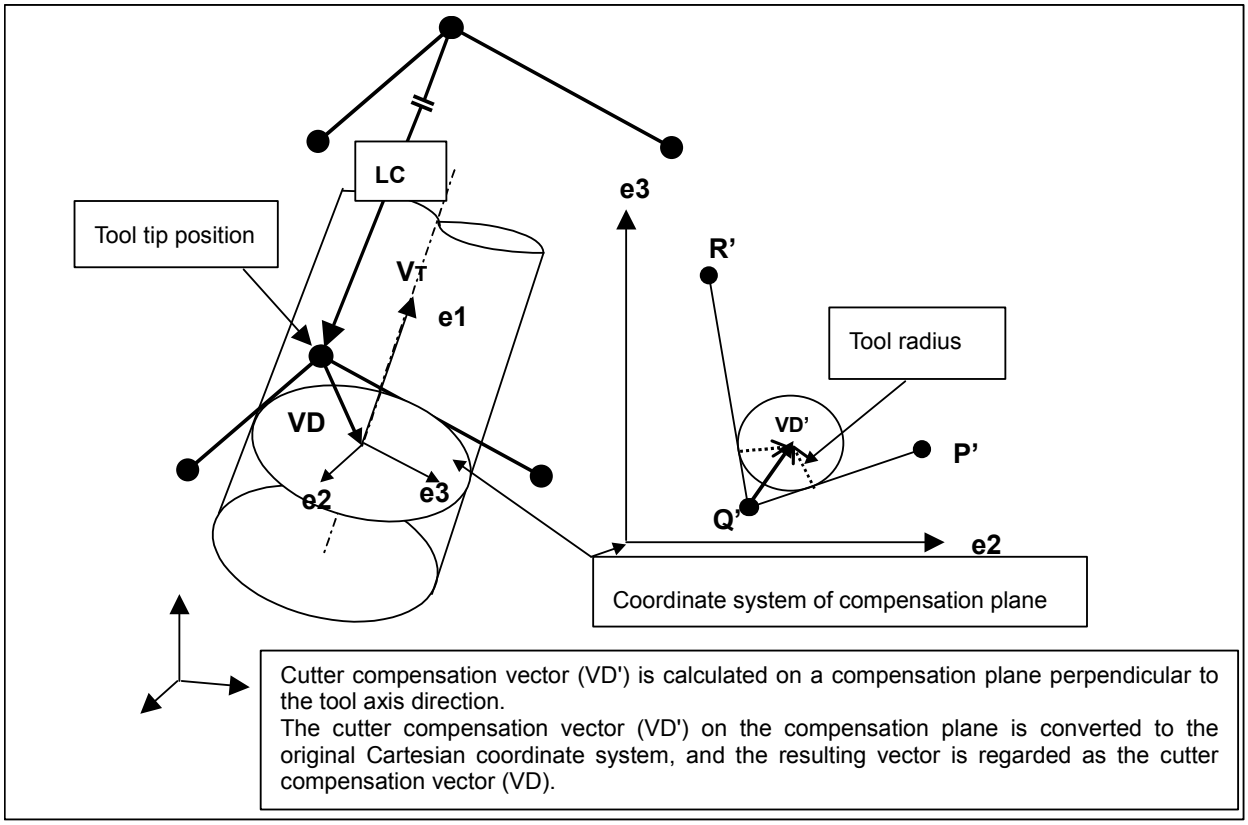


Fig. 22.8.1.3 (b) Operation

NOTE
1 This function is disabled for leading edge offset.
2 With a command for a rotary axis only, this function does not calculate a cutter compensation vector.
3 This function cannot be used in the 3-dimensional coordinate system conversion mode.
4 In addition to the cautions given here, the cautions on the 3-dimensional cutter compensation function apply to this function.

22.8.2 Cutter Compensation in Table Rotation Type Machine

Overview

Cutter compensation can be performed for a 5-axis machine having a rotary table as shown in Fig. 22.8.2 (a).

Shown below is a 5-axis machine that has table rotation axis A on the X-axis and table rotation axis B on the Y-axis.

This machine configuration is used as a sample configuration in the following explanation unless otherwise noted:

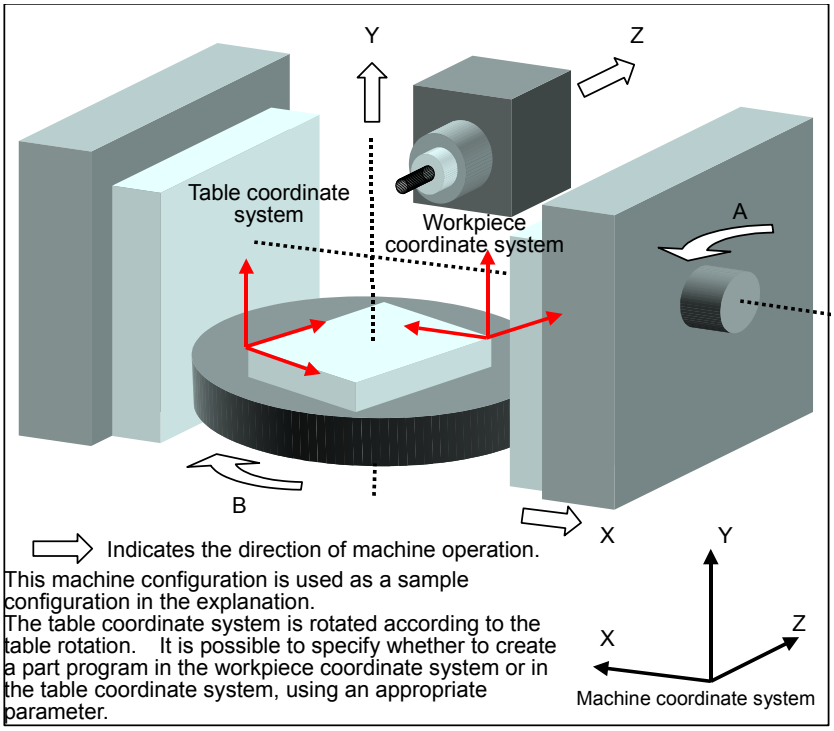


Fig. 22.8.2 (a) Machine having a rotary table

Format

- Startup (start of cutter compensation) (type 1)

When bit 1 (SPG) of parameter No. 19607 is 0

G41.2 (or G42.2) IP_ D_ ;

G41.2 : Cutter compensation left (group 07)
G42.2 : Cutter compensation right (group 07)
IP_ : Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)
D_ : Code specifying the cutter compensation amount (1 to 3 digits)

When bit 1 (SPG) of parameter No. 19607 is 1

G41.4 (or G42.4) IP_ D_ ;

G41.4 : Cutter compensation left (group 07)
G42.4 : Cutter compensation right (group 07)
IP_ : Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)
D_ : Code specifying the cutter compensation amount (1 to 3 digits)

NOTE
1 In a table rotation type machine (parameter No. 19680 = 12), if an attempt is made to issue G41.4 or G42.4 with bit 1 (SPG) of parameter No. 19607 equal to 0, alarm PS0010 is generated.
2 In a table rotation type machine, if an attempt is made to issue G41.2 or G42.2 with bit 1 (SPG) of parameter No. 19607 equal to 1, alarm PS5460 is generated.
3 In machine not of the table rotation type machine, if an attempt is made to issue G41.4 or G42.4 with bit 1 (SPG) of parameter No. 19607 equal to 1, alarm PS5460 is generated.

- Startup (start of cutter compensation) (type 2)

G41.6(or G42.6) IP_ D_ Q_ ;
IP_ I_ J_ K_ ;
:

G41.6 : Cutter compensation left (group 07)
G42.6 : Cutter compensation right (group 07)
IP_ : Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)
D_ : Code specifying the cutter compensation amount (1 to 3 digits)
Q_ : Tool's angle of gradient (in degrees)
I_ J_ K_ : Tool axis direction at the block end point as viewed from the programming coordinate system

For type 2, do not specify a rotation axis but specify the direction at the tool end point as viewed from the programming coordinate system (table coordinate system), with I, J, and K. Specifying a rotation axis causes alarm PS5460 to be generated.

In a tool rotation type machine, I, J, and K can be specified in a G41.6/G42.6 command block; in a table rotation type machine, however, they cannot. If an attempt is made to specify them, alarm PS5460 is generated.
The following are the notes on type 2.

NOTE
1 If one or two of I, J, and K are omitted, the omitted ones of I, J, and K are assumed to be 0.
2 In a block in which all of I, J, and K are omitted, the values of I, J, and K in the previous block are used.
3 If there is only one rotation axis (a hypothetical axis is used), type 2 cannot be used. In this case, if an attempt is made to issue G41.6/G42.6, alarm PS5460 is generated.
4 If using the rotation axis rollover function or rotary axis control function, specify 360 degrees in parameter No. 1260 (mount of travel per rotation about the rotation axis).
5 They can be used only with the settings that select the table coordinate system as a programming coordinate system (bit 5 (WKP) of parameter No.19696 = 0 and bit 4 (TBP) of parameter No.19746 = 1). If an attempt is made to issue G41.6/G42.6 with the settings that select the workpiece coordinate system as a programming coordinate system, alarm PS5460 is generated.

- Canceling the cutter compensation

G40 IP_ ;

G40: Cutter compensation cancellation (group 07)
IP_: Value specified for axis movement

- Selecting an offset plane

When bit 1 (PTD) of parameter No. 19746 is 1, compensation is performed on the selected plane, on the assumption that the tool is pointing to the direction perpendicular to that plane.

Table 22.8.2 (a)

Offset plane	Plane selection command	IP_
XpYp	G17 ;	Xp_Yp_
ZpXp	G18 ;	Xp_Zp_
YpZp	G19 ;	Yp_Zp_

(Example: In the case of Fig. 22.8.2 (a), the XpYp plane is selected.)
The two axes of a selected plane must be included in the three basic axes (the axes for which parameter No. 1022 is set to 1 to 3).

When bit 1 (PTD) of parameter No. 19746 is 0, compensation is performed on the plane perpendicular to the tool direction specified with parameters Nos. 19697, 19698, and 19699, regardless of which plane is selected.

NOTE
This function is enabled for tool side offset only. If leading edge offset is specified, alarm PS5460 is generated.

Explanation

- Tool's angle of gradient in type 2

For type 2 of 3-dimensional cutter compensation, the tool's angle of gradient can be specified with address Q in a G41.6/G42.6 command block. The tool's angle of gradient refers to the angle by which the tool direction to be assumed when machining is actually performed is inclined from the direction specified with (I, J, K) toward the traveling direction on the plane formed by the tool direction specified with (I, J, K) and the traveling direction in the programming coordinate system. (See Fig. 22.8.2 (b).)
Because in general, the normal direction of the machining surface is specified with (I, J, K), if it is desired to incline the tool direction to be assumed when machining is actually performed from the normal direction toward the traveling direction, correction may be performed with a Q command.
If the direction specified with (I, J, K) matches the tool direction to be assumed when machining is actually performed, no Q command is necessary.

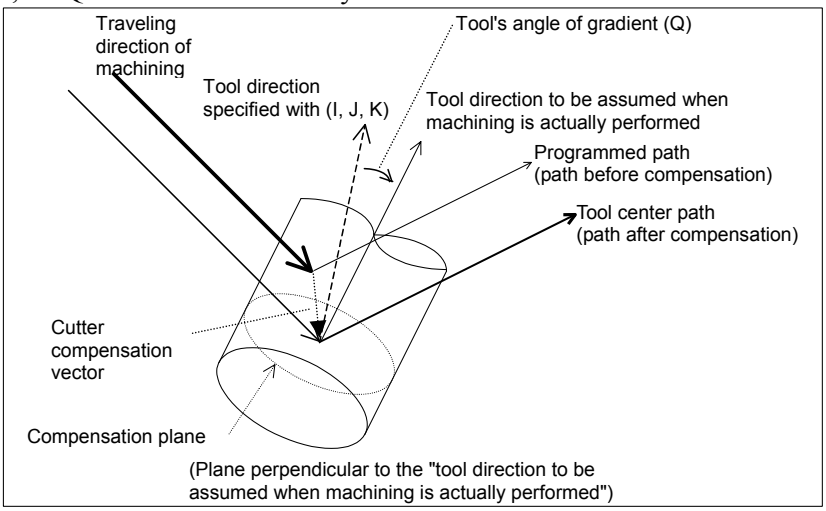


Fig. 22.8.2 (b) Tool's angle of gradient in type 2

- Cutter compensation

The function for 3-dimensional cutter compensation in a table rotation type machine basically performs operations in conformance with 3-dimensional cutter compensation in a tool rotation type machine. The operations different from those of 3-dimensional cutter compensation in a tool rotation type machine are mainly described below.
For the specifications and cautions not mentioned here, see the description of 3-dimensional cutter compensation in a tool rotation type machine.

- Startup

When cutter compensation for the rotary table is specified (G41.2 or G42.2, G41.4 or G42.4, a dimension word other than 0 in the offset plane, or a D code other than D0) in the offset cancel mode, the CNC enters the offset mode.
Startup is specified with positioning (G00) or linear interpolation (G01).

NOTE
If a command such as circular interpolation (G02 or G03) and involute interpolation (G02.2 or G03.2) is specified at startup, alarm PS0034 is issued.

- Commands in the offset mode

In the offset mode, compensation is performed for positioning (G00) and linear interpolation (G01).

NOTE
If a command such as circular interpolation (G02 or G03) and involute interpolation (G02.2 or G03.2) is specified in the mode for 3-dimensional cutter compensation in a table rotation type machine, alarm PS5460 is issued.

- Offset mode cancellation

If a block satisfying either of the following conditions is executed in the offset mode, the CNC enters the offset cancel mode:
1 G40 is specified.
2 0 is specified as a code specifying the cutter compensation amount (D code).
When offset cancellation is to be performed, neither circular (G02 or G03) nor involute (G02.2 or G03.2) command can be specified. If offset cancellation is specified in such a mode, alarm PS0034 is issued.

- If selecting the table coordinate system as a programming coordinate system

If bit 4 (TBP) of parameter No. 19746 is 1 and bit 5 (WKP) of parameter No. 19696 is 0, specifying 3-dimensional cutter compensation causes the table coordinate system to be selected as a programming coordinate system. The table coordinate system refers to the workpiece coordinate system fixed to the table when 3-dimensional cutter compensation.
In the blocks subsequent to the one in which 3-dimensional cutter compensation is specified, the table coordinate system rotates with the rotation of the table.
A linear axis (X, Y, Z) command is assumed to be issued on the table coordinate system.
Specifying linear interpolation causes cutter compensation to be performed on the linear interpolation command on the table coordinate system.
The table coordinate system does not rotate with the rotation of the tool head.

The cancel (G40) block becomes a block that suppresses buffering.

It is possible to switch between absolute and relative coordinate displays, using bit 2 (DET) of parameter No. 19608 as follows:

- When DET is 0, the position in the table coordinate system is displayed.
- When DET is 1, the position in the workpiece coordinate system is displayed.

Note, however, the distance to go is always that in the programming coordinate system.

NOTE

1 If the 3-dimensional cutter compensation mode is entered when the table coordinate system is used as the programming coordinate system, look-ahead acceleration/deceleration before interpolation is automatically enabled. Be sure to specify the following parameters:
   (1) Bit 1 (LRP) of parameter No.1401 = 1:
   Linear rapid traverse
   (2) Bit 5 (FRP) of parameter No.19501 = 1:
   Acceleration/deceleration before interpolation is used for rapid traverse.
   (3) Parameter No. 1671:
   Acceleration of acceleration/deceleration before interpolation for rapid traverse
   (4) Parameter No. 1672:
   Change time for bell-shaped acceleration/deceleration before interpolation for rapid traverse
   (5) Parameter No. 1660:
   Maximum permissible acceleration for acceleration/deceleration before interpolation
   If they are not specified, alarm PS5463 is generated.
2 When table rotation axis movement is specified in the start block of 3-dimensional cutter compensation, after the movement is completed, the workpiece coordinate system is fixed to the table and assumed to be a table coordinate system.
3 In the mode of 3-dimensional cutter compensation, do not change the workpiece coordinate system or change the workpiece offset value. If an attempt is made to specify workpiece coordinate system selection (G54 to G59), alarm PS5460 is generated.

- If selecting the workpiece coordinate system as a programming coordinate system

If bit 4 (TBP) of parameter No. 19746 is 0 or if bit 4 (TBP) of parameter No. 19746 is 1 and bit 5 (WKP) of parameter No. 19696 is 1, the programming coordinate system does not rotate with the rotation of the table, being fixed to the workpiece coordinate system.

- Angle of the rotation axis in type 2

For an explanation of how the rotation axis end point is determined when the tool direction is specified with an IJKQ command in type 2, see the explanations of cutter compensation in a tool rotation type machine, "Angle of the rotation axis in type 2 (if the operating range is not specified)" and "Angle of the rotation axis in type 2 (if the operating range is specified)".

22.8.3 Cutter Compensation in Composite Type Machine

Overview

This function can perform 3-dimensional cutter compensation in a 5-axis machine having a rotary table and a tool axis as shown in Fig. 22.8.3 (a).

Shown below is a 5-axis machine that has tool axis A on the X-axis (the tool axis direction is along the Z-axis) and table rotation axis B on the Y-axis.

This machine configuration is used as a sample configuration in the following explanation unless otherwise noted:

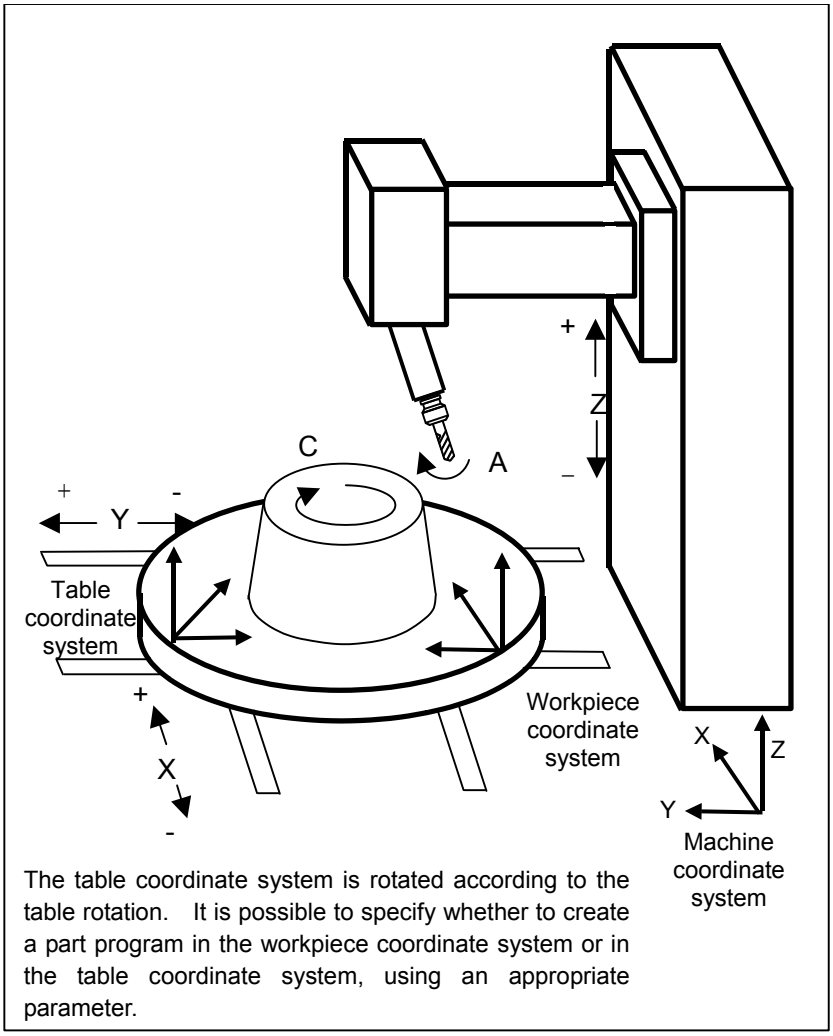


Fig. 22.8.3 (a) Machine having a tool rotation axis and table rotation axis

Format

- Startup (start of cutter compensation) (type 1)

When bit 1 (SPG) of parameter No. 19607 is 0

G41.2 (or G42.2) IP_ D_ ;

G41.2 : Cutter compensation left (group 07)
G42.2 : Cutter compensation right (group 07)
IP_ : Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)
D_ : Code specifying the cutter compensation amount (1 to 3 digits)

When bit 1 (SPG) of parameter No. 19607 is 1

G41.5 (or G42.5) IP_ D_ ;

G41.5 : Cutter compensation left (group 07)
G42.5 : Cutter compensation right (group 07)
IP_ : Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)
D_ : Code specifying the cutter compensation amount (1 to 3 digits)

NOTE
1 In a composite type machine (parameter No. 19680 = 21), if an attempt is made to issue G41.5 or G42.5 with bit 1 (SPG) of parameter No. 19607 equal to 0, alarm PS0010 is generated.
2 In a composite type machine, if an attempt is made to issue G41.2 or G42.2 with bit 1 (SPG) of parameter No. 19607 equal to 1, alarm PS5460 is generated.
3 In machine not of the composite type machine, if an attempt is made to issue G41.5 or G42.5 with bit 1 (SPG) of parameter No. 19607 equal to 1, alarm PS5460 is generated.

- Startup (start of cutter compensation) (type 2)

G41.6 (or G42.6) IP_ D_ Q_ ;
IP_ I_ J_ K_ ;
:

G41.6 : Cutter compensation left (group 07)
G42.6 : Cutter compensation right (group 07)
IP_ : Value specified for axis moving as viewed from the programming coordinate system (including rotary axis)
D_ : Code specifying the cutter compensation amount (1 to 3 digits)
Q_ : Tool's angle of gradient (in degrees)
I_ J_ K_ : Tool axis direction at the block end point as viewed from the programming coordinate system

For type 2, do not specify a rotation axis but specify the direction at the tool end point as viewed from the programming coordinate system (table coordinate system), with I, J, and K. Specifying a rotation axis causes alarm PS5460 to be generated.

In a tool rotation type machine, I, J, and K can be specified in a G41.6/G42.6 command block; in a composite type machine, however, they cannot. If an attempt is made to specify them, alarm PS5460 is generated.
The following are the notes on type 2.

NOTE
1 If one or two of I, J, and K are omitted, the omitted ones of I, J, and K are assumed to be 0.
2 In a block in which all of I, J, and K are omitted, the values of I, J, and K in the previous block are used.
3 If there is only one rotation axis (a hypothetical axis is used), type 2 cannot be used. In this case, if an attempt is made to issue G41.6/G42.6, alarm PS5460 is generated.
4 If using the rotation axis rollover function or rotary axis control function, specify 360 degrees in parameter No. 1260 (mount of travel per rotation about the rotation axis).

NOTE
5 They can be used only with the settings that select the table coordinate system as a programming coordinate system (bit 5 (WKP) of parameter No.19696 = 0 and bit 4 (TBP) of parameter No.19746 = 1). If an attempt is made to issue G41.6/G42.6 with the settings that select the workpiece coordinate system as a programming coordinate system, alarm PS5460 is generated.

- Canceling the cutter compensation

G40 IP_ ;
G40 : Cutter compensation cancellation (group 07)
IP_ : Value specified for axis movement

NOTE
This function is enabled for tool side offset only. If leading edge offset is specified, alarm PS5460 is generated.

Explanation

- Tool's angle of gradient in type 2

For type 2 of 3-dimensional cutter compensation, the tool's angle of gradient can be specified with address Q in a G41.6/G42.6 command block. The tool's angle of gradient refers to the angle by which the tool direction to be assumed when machining is actually performed is inclined from the direction specified with (I, J, K) toward the traveling direction on the plane formed by the tool direction specified with (I, J, K) and the traveling direction in the programming coordinate system. (See Fig. 22.8.3 (b).)
Because in general, the normal direction of the machining surface is specified with (I, J, K), if it is desired to incline the tool direction to be assumed when machining is actually performed from the normal direction toward the traveling direction, correction may be performed with a Q command.
If the direction specified with (I, J, K) matches the tool direction to be assumed when machining is actually performed, no Q command is necessary.

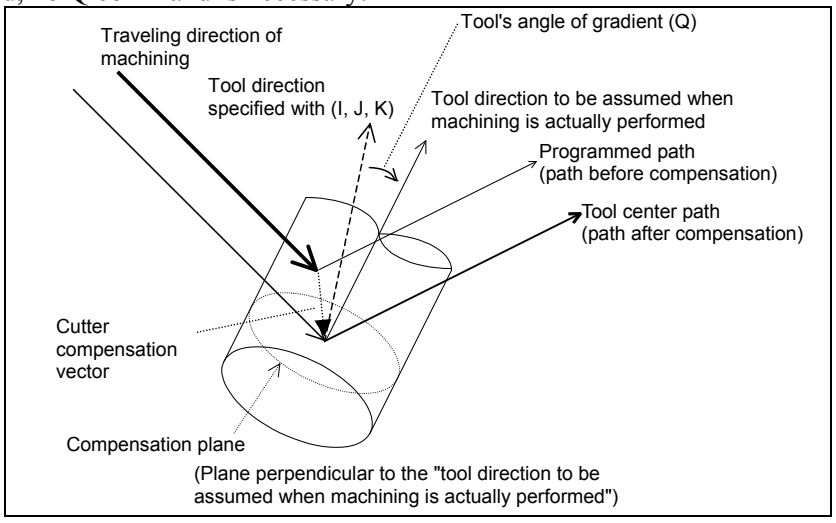


Fig. 22.8.3 (b) Tool's angle of gradient in type 2

- Cutter compensation

The function for 3-dimensional cutter compensation in a composite type machine basically performs operations in conformance with 3-dimensional cutter compensation in a tool rotation type machine. The operations different from those of 3-dimensional cutter compensation in a tool rotation type machine are mainly described below.
For the specifications and cautions not mentioned here, see the description of 3-dimensional cutter compensation in a tool rotation type machine.

- Startup

When 3-dimensional cutter compensation in a composite type machine (G41.2 or G42.2, G41.5 or G42.5, or a D code other than D0) is specified in the offset cancel mode, the CNC enters the offset mode.
Startup is specified with positioning (G00) or linear interpolation (G01).

NOTE
If a command such as circular interpolation (G02 or G03) and involute interpolation (G02.2 or G03.2) is specified at startup, alarm PS0034 is issued.

- Commands in the offset mode

In the offset mode, compensation is performed for positioning (G00) and linear interpolation (G01).

NOTE
If a command such as circular interpolation (G02 or G03) and involute interpolation (G02.2 or G03.2) is specified in the mode for 3-dimensional cutter compensation in a composite type machine, alarm PS5460 is issued.

- Offset mode cancellation

If a block satisfying either of the following conditions is executed in the offset mode, the CNC enters the offset cancel mode:

1 G40 is specified.
2 0 is specified as a code specifying the cutter compensation amount (D code).

When offset cancellation is to be performed, neither circular (G02 or G03) nor involute (G02.2 or G03.2) command can be specified. If offset cancellation is specified in such a mode, alarm PS0034 is issued.

NOTE
1 This function cannot be used in the 3-dimensional coordinate system conversion mode.
2 In addition to the cautions given here, the cautions on the 3-dimensional cutter compensation function apply to this function.

- If selecting the table coordinate system as a programming coordinate system

If bit 4 (TBP) of parameter No.19746 is 1 and bit 5 (WKP) of parameter No.19696 is 0, specifying 3-dimensional cutter compensation causes the table coordinate system to be selected as a programming coordinate system. The table coordinate system refers to the workpiece coordinate system fixed to the table when 3-dimensional cutter compensation.
In the blocks subsequent to the one in which 3-dimensional cutter compensation is specified, the table coordinate system rotates with the rotation of the table.
A linear axis (X, Y, Z) command is assumed to be issued on the table coordinate system.
Specifying linear interpolation causes cutter compensation to be performed on the linear interpolation command on the table coordinate system.
The table coordinate system does not rotate with the rotation of the tool head.

The cancel (G40) block becomes a block that suppresses buffering.

It is possible to switch between absolute and relative coordinate displays, using bit 2 (DET) of parameter No.19608 as follows:

- When DET is 0, the position in the table coordinate system is displayed.
- When DET is 1, the position in the workpiece coordinate system is displayed.

Note, however, the distance to go is always that in the programming coordinate system.

NOTE

1 If the 3-dimensional cutter compensation mode is entered when the table coordinate system is used as the programming coordinate system, look-ahead acceleration/deceleration before interpolation is automatically enabled. Be sure to specify the following parameters:
   (1) Bit 1 (LRP) of parameter No.1401 = 1:
   Linear rapid traverse
   (2) Bit 5 (FRP) of parameter No.19501 = 1:
   Acceleration/deceleration before interpolation is used for rapid traverse.
   (3) Parameter No. 1671:
   Acceleration of acceleration/deceleration before interpolation for rapid traverse
   (4) Parameter No. 1672:
   Change time for bell-shaped acceleration/deceleration before interpolation for rapid traverse
   (5) Parameter No. 1660:
   Maximum permissible acceleration for acceleration/deceleration before interpolation
   If they are not specified, alarm PS5463 is generated.
2 When table rotation axis movement is specified in the start block of 3-dimensional cutter compensation, after the movement is completed, the workpiece coordinate system is fixed to the table and assumed to be a table coordinate system.
3 In the mode of 3-dimensional cutter compensation, do not change the workpiece coordinate system or change the workpiece offset value. If an attempt is made to specify workpiece coordinate system selection (G54 to G59), alarm PS5460 is generated.

- If selecting the workpiece coordinate system as a programming coordinate system

If bit 4 (TBP) of parameter No.19746 is 0 or if bit 4 (TBP) of parameter No.19746 is 1 and bit 5 (WKP) of parameter No.19696 is 1, the programming coordinate system does not rotate with the rotation of the table, being fixed to the workpiece coordinate system.

- Angle of the rotation axis in type 2

For an explanation of how the rotation axis end point is determined when the tool direction is specified with an IJKQ command in type 2, see the explanations of cutter compensation in a tool rotation type machine, "Angle of the rotation axis in type 2 (if the operating range is not specified)" and "Angle of the rotation axis in type 2 (if the operating range is specified)".

22.8.4 Interference Check and Interference Avoidance

Overview

By setting bit 1 (NI5) of parameter No. 19608 to 1, this function performs an interference check on the plane (compensation plane) perpendicular to the tool axis direction regardless of the machine configuration.
If bit 5 (CAV) of parameter No. 19607 is set to 1, a vector is generated to avoid interference on the same plane.

Explanation

- For a tool rotation type machine

An interference check is performed, as well as interference avoidance, with the tool path as projected from the workpiece coordinate system (X-Y-Z) onto the compensation plane (X'-Y'-Z') and a compensation vector.

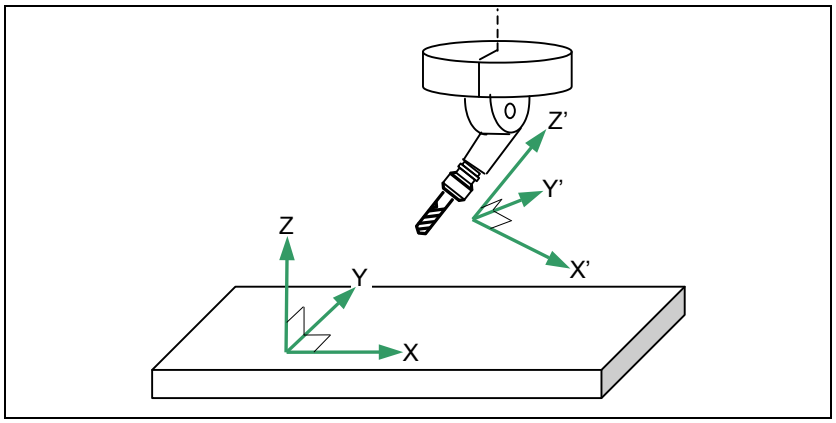


Fig. 22.8.4 (a) Tool rotation type

- For a table rotation type machine

An interference check is performed, as well as interference avoidance, with the tool path as converted from the workpiece coordinate system (X-Y-Z) into the table coordinate system (X'-Y'-Z') and a compensation vector.

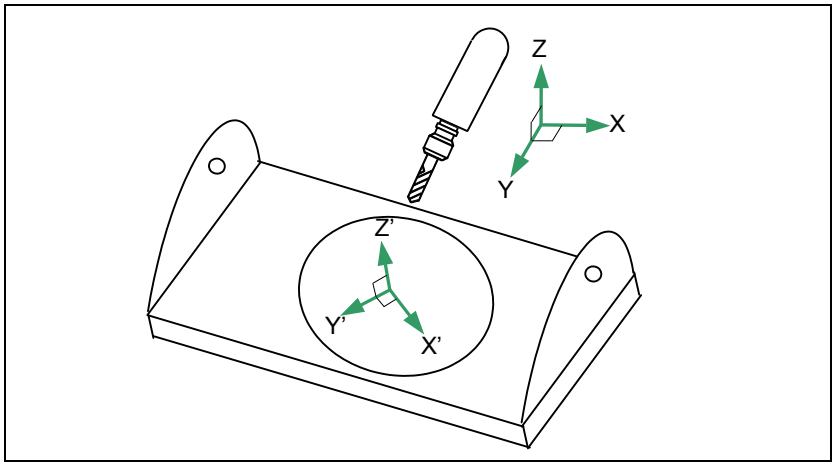


Fig. 22.8.4 (b) Table rotation type

- For a composite type machine

An interference check is performed, as well as interference avoidance, with the tool path as projected from the workpiece coordinate system (X-Y-Z) onto the table coordinate system (X'-Y'-Z') and then onto the compensation plane perpendicular to the tool axis direction (X"-Y"-Z") and a compensation vector.

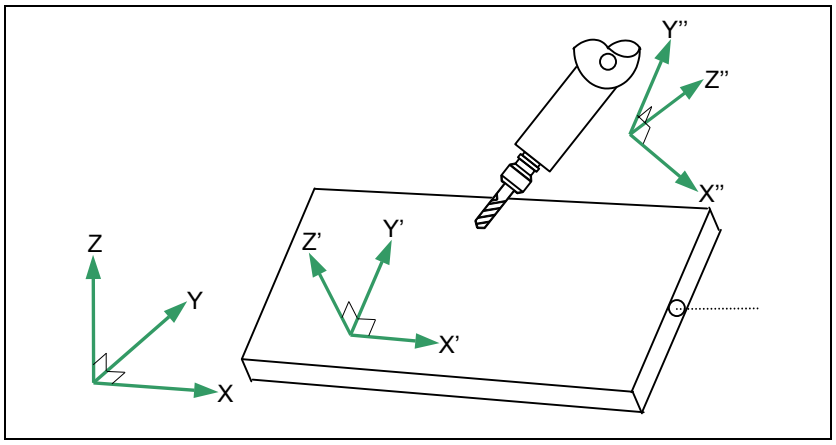


Fig. 22.8.4 (c) Composite type

- **Interference avoidance**

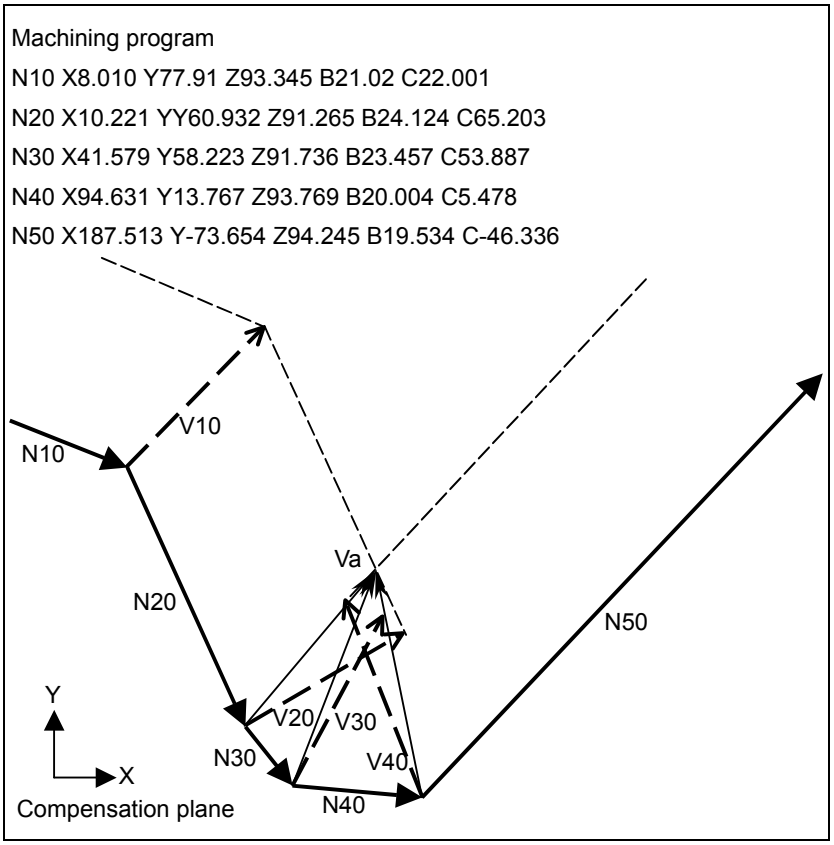


Fig. 22.8.4 (d) Example of interference avoidance

Fig. 22.8.4 (d) shows a tool path in the workpiece coordinate system as projected onto the compensation plane. This is an example when the parameter No.19626 is set to 4.
At the start of the execution of the N10 block, the system looks at N20 to N50 ahead and generates V20 to V40.
Then, because the movement direction of N30 greatly differs from the direction from V20 to V30, V20 and V30 are considered interferences and eliminated. Similarly, because the movement direction from N30 to N40 greatly differs from the movement direction from V20 to V40, V40 is also considered an interference and eliminated.
Then, the interference avoidance vector Va is generated between N20 and N50 and uses it instead of V20, V30, and V40. At this time, because N20 and N50 generally do not intersect, the plane perpendicular to the tool axis direction at the N20 end point is regarded as a compensation plane, N20 and N50 are projected onto this plane, and Va is determined by calculating the intersection point.

NOTE

Strictly speaking, if the tool axis direction at the N20 end point differs from the tool axis direction at the N50 start point, correct intersection point calculation is not possible. For this reason, the maximum permissible angle by which the tool axis directions in the two blocks used to determine the interference avoidance vector may differ can be set in parameter No. 19636, and if the change in tool axis direction is within the range, an approximate interference avoidance vector is calculated.

If the maximum angle is exceeded, the compensation vector is determined but interference avoidance is not performed.

- If interference avoidance is not possible

If there are three consecutive interfering blocks, no interference vector can be generated.

Example 1 in which interference avoidance is not possible

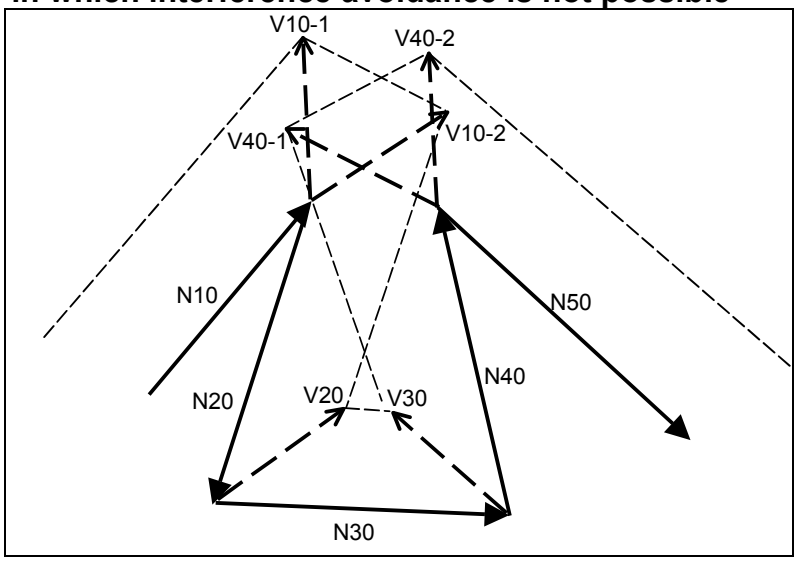


Fig. 22.8.4 (e)

N20 to N40 interfere, so that no interference avoidance vector can be generated. Too much cutting results.

Example 2 in which interference avoidance is not possible

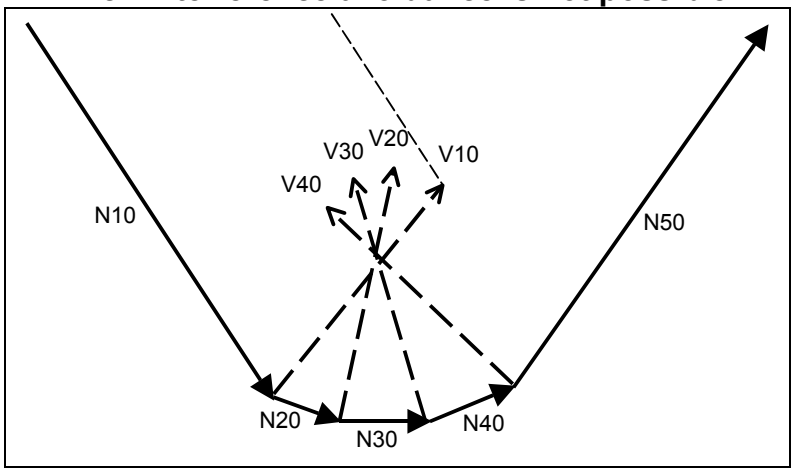


Fig. 22.8.4 (f)

N10 to N40 interfere, so that no interference avoidance vector can be generated. V10 causes an interference alarm.

22.8.5 Restrictions

22.8.5.1 Restrictions common to machine configurations

- Interference check

In the 3-dimensional tool radius compensation, an interference check is performed using a specified position in the workpiece coordinate system and a compensation vector. The interference check avoidance function cannot be used.

- Corner rounding (G39)

In the mode for 3-dimensional cutter compensation, G39 cannot be specified. Specifying G39 causes an alarm.

- Reset

Whenever a reset is made in the mode for 3-dimensional cutter compensation (G41.2, G42.2, G41.4, G42.4, G41.5, or G42.5), the cancel mode (G40) is entered.

- AI contour control I and II

To set the AI contour control I or II mode, the corresponding G code must be specified. Specifying 3-dimensional cutter compensation does not automatically place the CNC in the AI contour control I or II mode.

- Restricted commands

In the mode for 3-dimensional cutter compensation, the following functions are available, but their statuses cannot be changed:

- Inch/metric input (If an attempt is made to change the status by using G20 or G21, alarm PS5000 is issued.)
- Mirror image (The signal status cannot be changed.)
- One-digit F code feed (The feedrate cannot be changed by using the manual handle.)

- When a rotary axis is moved significantly

In 3-dimensional cutter compensation, compensation is performed at the block end point.
Even when the tool direction relative to the workpiece changes momentarily as a rotary axis moves, compensation is not performed momentarily during execution of the block.
Accordingly, when a rotary axis is moved significantly, carry out programming to divide a block into small movement blocks to prevent the machining precision from being reduced.

- Operation at cancellation

In a block for canceling 3-dimensional cutter compensation, compensation accompanying movement is cancelled regardless of whether the movement command is present.

- Use with tool center point control

If bit 1 (TCC) of parameter No.11269 is set to 1, G41/G42 command in tool center point control mode is handled as 3-dimensional cutter compensation (tool side offset). G code of 3-dimensional cutter compensation depends on type of tool center point control, parameter, and machine configuration as indicated in following table.

Command G code	Tool center point control	Bit 1 (SPG) of parameter No.19607	Machine configuration	Internal G code
G41 / G42	Type 1	0	-	G41.2 / G42.2
		1	Tool rotation type	G41.2 / G42.2
			Table rotation type	G41.4 / G42.4
			Composite type	G41.5 / G42.5
	Type 2	-	-	G41.6 / G42.6

NOTE
1 The leading edge offset is not supported.
2 If bit 1 (TCC) of parameter No.11269 is set to 1 and the 3-dimensional cutter compensation option is not active, alarm PS5463 is issued when G41/G42 is specified in tool center point control mode.
3 If bit 1 (TCC) of parameter No.11269 is set to 1, cutter compensation and tool nose radius compensation, 3-dimensional tool compensation cannot be specified.

- Unavailable commands

In the mode for 3-dimensional cutter compensation, the functions listed below cannot be specified. Specifying any of these functions results in an alarm.

- Hypothetical axis interpolation G07
- Circular interpolation G07.1
- Polar coordinate interpolation G12.1, G13.1
- Polar coordinate command G15, G16
- Reference position return check G27
- Reference position return G28, G29, G30
- Skip G31
- Threading G33
- Automatic tool length measurement G37
- Normal direction control G40.1, G41.1, G42.1
- Tool radius/tool nose radius compensation G41, G42, G39
- 3-dimensional cutter compensation G41
- Wheel wear compensation G41
- Tool offset G45, G46, G47, G48
- Local coordinate system G52
- Machine coordinate system G53
- Workpiece coordinate system setting G54-G59, G54.1
- Rotary table dynamic fixture offset G54.2
- Single direction positioning G60
- Automatic corner override G62
- Tapping mode G63
- 3-dimensional coordinate system conversion G68, G69
- Tilted working plane indexing G68.2, G69
- Figure copy G72.1, G72.2
- Canned cycle G73-G79, G80, G81-G89, G98, G99
- Electric gear box G80, G81
- Hobbling machine function G80, G81
- External operation function G81
- Chopping G81.1
- Small hole peck drilling cycle G83
- Workpiece coordinate system setting G92
- Workpiece coordinate system presetting G92.1

- Feed per revolution G95
- Constant surface speed control G96, G97

- Unavailable functions

If the following function is specified in the 3-dimensional cutter compensation mode, a warning message is issued:

- MDI interruption

If one of the following functions is specified in the 3-dimensional cutter compensation mode, a PS alarm is issued:

- Manual interruption operation
- Tool retract and recover

In the 3-dimensional cutter compensation mode, the following functions cannot be used:

- Twin table control
- Flexible synchronous control
- Sequence number comparison and stop (It is not possible to cause a stop by sequence number in the 3-dimensional cutter compensation mode.)
- Index table indexing
- Retrace function (Programs using the 3-dimensional cutter compensation mode must not use the retrace function.)
- Rotary axis control
- Manual handle interruption
- External deceleration (External deceleration is not performed.)
- Angular axis control

Combinations of other NC commands may be restricted. Refer to the manual on each function.

22.8.5.2 Restriction on tool rotation type

- Unavailable commands (leading edge offset)

In the G41.3 mode, the following commands cannot be specified:

- G functions of group 01 other than G00 and G01

- Use with tool center point control

If 3-dimensional cutter compensation is specified before tool center point control when 3-dimensional cutter compensation and tool center point control are specified together, tool center point control must be canceled before 3-dimensional cutter compensation is canceled (Example 1). Contrariwise, if tool center point control is specified before 3-dimensional cutter compensation, 3-dimensional cutter compensation must be canceled before tool center point control is canceled (Example 2).

```
Example 1

G41.2   D1
 :
G43.4   H1
 :
G49
 :
G40
```

```
Example 2

G43.4   H1
 :
G41.2   D1
 :
G40
 :
G49
```

Fig. 22.8.5.2 (a)

If the specification sequence of G40 and G49 is reversed, alarm PS5460 is issued.

When 3-dimensional cutter compensation is specified before tool center point control, the block for canceling tool center point control suppresses buffering. Note that, as a result, the block before the G49 block generates a compensation vector for 3-dimensional cutter compensation, which is perpendicular to the movement.
For each of 3-dimensional cutter compensation and tool center point control, two commands, type 1 and type 2, are provided. Be sure to specify commands of the same type. If commands of different types are specified, alarm PS5460 is issued.
When the tool is tilted by address Q in type 2 command specification, if Q is specified in both of the block for starting tool center point control and the block for starting 3-dimensional cutter compensation, the Q command specified earlier becomes valid.

- Combination with circular interpolation

When circular interpolation is specified in a block, if a rotary axis is moved in the block, the previous block, or the next block, a cutter compensation vector may be created in the direction orthogonal to the plane. In this case, helical interpolation is performed, so the helical interpolation option is required.

22.8.5.3 Restriction on machine configurations having table rotation axes (table rotation type and composite type)

- Unavailable commands

For machines having table rotation axes, the following commands cannot be specified during 3-dimensional cutter compensation:

- G functions of group 01 other than G00 and G01

- Use with tool center point control

When the workpiece coordinate system is used as the programming coordinate system, the same restrictions as for the tool rotation type apply. See "Use with tool center point control" in the restrictions on the tool rotation type described previously.
When the table coordinate system is used as the programming coordinate system, the restrictions to apply will be explained below.

- Type 2

When a type 2 command (G41.6/G42.6 command) is executed, the table coordinate system needs to be used as the programming coordinate system (bit 4 (TBP) of parameter No. 19746 = 1 and bit 5 (WKP) of parameter No. 19696 = 0). If a type 2 command (G41.6/G42.6 command) is executed with the workpiece coordinate system used as the programming coordinate system, alarm PS5459 is issued.

- Restrictions when the table coordinate system is used as the programming coordinate system

Use with tool center point control

When 3-dimensional cutter compensation and tool center point control are specified together, tool center point control must be specified before cutter compensation. In addition, tool center point control must be canceled after 3-dimensional cutter compensation is canceled. (Example 1)
If tool center point control is specified in the 3-dimensional cutter compensation mode (Example 2), or if tool center point control is canceled without canceling 3-dimensional cutter compensation (Example 3), alarm PS5460 is issued.

Example 1
(Correct specification)

```
G43.4  H1
 :
G41.2  D1
 :
G40
 :
G49
```

Example 2
(Specification resulting in alarm)

```
G41.2  D1
 :
G43.4  H1
   :
```

Example 3
(Specification resulting in alarm)

```
G43.4  H1
 :
G41.2  D1
 :
G49
```

Fig. 22.8.5.3 (a)

If the setting of the programming coordinate system differs between 3-dimensional cutter compensation and tool center point control, specifying both functions together results in alarm PS5460. (See the following table:)

Table 22.8.5.3 (a)

	TBP=0		TBP=1	
	WKP=0	WKP=1	WKP=0	WKP=1
Tool center point control	Table coordinate system	Workpiece coordinate system	Table coordinate system	Workpiece coordinate system
3-dimensional cutter compensation	Workpiece coordinate system			
Both specified together	Alarm PS5460			

Programming coordinate system determined by bit 4 (TBP) of parameter No. 19746 and bit 5 (WKP) of parameter No. 19696

The table rotation axis position at the start of tool center point control and the table rotation axis position at the start of 3-dimensional cutter compensation must always match.
Each programming coordinate system matches the workpiece coordinate system used when each function is started.
If these functions are started when their table rotation axis positions differ, the programming coordinate system for tool center point control and the programming coordinate system for 3-dimensional cutter compensation do not match, which results in alarm PS5460.
Example: When the A-axis is the table rotation axis:
(Correct example)

```
G90 G00 A0.0
G43.4 H1
G01 Z100.0 F1000.0
G41.2 D1 ←        After G43.4 is specified, G41.2 is specified without A-axis movement.
:
```

(Wrong example)

```
G90 G00 A0.0
G43.4 H1
G01 Z100.0 A30.0 F1000.0
G41.2 D1 ←        After G43.4 is specified, A-axis movement is made, then G41.2 is specified.
:
```

* Furthermore, no A-axis command can be included in the block specifying G41.2.

For each of 3-dimensional cutter compensation and tool center point control, two commands, type 1 and type 2, are provided. Be sure to specify commands of the same type. If commands of different types are specified, alarm PS5460 is issued.
When the tool is tilted by address Q in type 2 command specification, if Q is specified in both of the block for starting tool center point control and the block for starting 3-dimensional cutter compensation, the Q command specified earlier becomes valid.

Look-ahead acceleration/deceleration before interpolation

If the 3-dimensional cutter compensation mode is entered, look-ahead acceleration/deceleration before interpolation is automatically enabled. Set the look-ahead acceleration/deceleration parameter. If the parameter is incorrectly set, an alarm is issued.

Deceleration at a corner

Under 3-dimensional cutter compensation, the controlled point may move along a curve even if a straight-line command is issued. Some commands may cause a corner movement.
When AI contour control is not used, there is a possibility that the machine is shocked at a corner.
Accordingly, it is recommended that AI contour control be used and the permissible acceleration (parameter No. 1737) and permissible speed difference at a corner (parameter No. 1783) be set.

Specifiable G codes

When the table coordinate system is used as the programming coordinate system, the G codes that can be specified in the 3-dimensional cutter compensation mode are listed below.
Specifying a G code other than these codes results in alarm PS5460:

- Positioning (G00)
- Linear interpolation (G01)
- Dwell (G04)
- Exact stop (G09)
- Programmable data input (G10)
- Programmable data input mode cancel (G11)
- Plane selection (G17/G18/G19)
- Cutter compensation : cancel (G40)
- Tool length compensation cancel (G49,G49.1)
- Programmable mirror image (G50.1/G51.1)
- Macro call (G65)
- Macro modal call A (G66)
- Macro modal call B (G66.1)
- Macro modal call A/B cancel (G67)
- Absolute programming (G90)
- Incremental programming (G91)

Modal G codes that allow specification of 3-dimensional cutter compensation

When the table coordinate system is used as the programming coordinate system, 3-dimensional cutter compensation can be specified in the modal G code states listed below.
In a modal state other than the following modal G codes, specifying tool center point control results in alarm PS5421.

- Modal G codes included in "Specifiable G codes" described previously
- Polar coordinate interpolation mode cancel (G13.1)
- Polar coordinates command cancel (G15)
- Input in inch (G20 (G70))
- Input in mm (G21 (G71))
- Tool center point control (G43.4/G43.5)
- Polygon turning cancel (G50.2)
- Workpiece coordinate system 1 selection (G54 to G59)
- Canned cycle cancel (G80)
- Constant surface speed control cancel (G97)
- Canned cycle : return to initial level (G98)
- Canned cycle : return to R point level (G99)

M

- Coordinate system rotation start or 3-dimensional coordinate conversion mode on (G69)
- Feed per minute (G94)
- Polar coordinate interpolation mode cancel (G113)

T

- Mirror image for double turret off/balanced cutting mode cancel (G69)
- Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off (G69.1)
- Feed per minute (G98 (G94))

Specification of axes not relating to 3-dimensional cutter compensation

Axes not relating to 3-dimensional cutter compensation cannot be specified. If such an axis is specified, alarm PS5460 is issued.

22.8.6 Examples

O100 is a sample program.
This is an example in which each side of a square is cut at an angle of 30 degrees on the B-axis in a composite type machine.
Programs 1 to 3 all perform the same machining.

Program 1: Type 1 and the table coordinate system is selected as a programming coordinate system

```
O100（Sample Program1）；
N10 G55 ;                              Preparations for the programming coordinate system
N20 G90 X0 Y0 Z300.0 B0 C0 ;           Movement to the initial position
N30 G01 G43.4 H01 Z40.0 F500.0 ;       Start of tool center point control
                                       H01 is a tool length compensation number
N40 G41.2 D01;                         Start of cutter compensation
                                       D01 is a tool cutter compensation number
N50 X50.0 Y50.0 Z20.0 B30.0 C45.0 ;    The Z-axis height on the machining plane is 20.0.
N60 X-50.0 C135.0 ;
N70 X-100.0 Y-100.0 C225.0 ;
N80 X100.0 C315.0 ;
N90 X50.0 Y50.0 C405.0 ;
N100 X0 Y0 Z40.0 B0 C360.0 ;
N110 G40 ;                             Cutter compensation cancellation
N120 G49 Z300.0 ;                      Tool center point control cancellation
                                       Movement to the initial position on the Z-axis
N130 M30;
```

Program 2: Type 1 and the workpiece coordinate system is selected as a programming coordinate system
(Note that the values specified in N50 to N90 differ from those in program 1.)

```
O100（Sample Program2）;
N10 G55 ;                          Preparations for the programming coordinate system
N20 G90 X0 Y0 Z300.0 B0 C0 ;       Movement to the initial position
N30 G01 G43.4 H01 Z40.0 F500.0 ;   Start of tool center point control
                                   H01 is a tool length compensation number
N40 G41.2 D01;                     Start of cutter compensation
                                   D01 is a tool cutter compensation number
N50 X70.711 Y0 Z20.0 B30.0 C45.0 ; The Z-axis height on the machining plane is 20.0.
N60 C135.0 ;
N70 X141.421 C225.0 ;
N80 C315.0 ;
N90 X70.711 C405.0 ;
N100 X0 Y0 Z40.0 B0 C360.0 ;
N110 G40 ;                         Cutter compensation cancellation
N120 G49 Z300.0 ;                  Tool center point control cancellation
                                   Movement to the initial position on the Z-axis
N130 M30;
```

Program 3: When the type 2 is used:
(The table coordinate system is selected as a programming coordinate system)

```
O100（Sample Program3）;
N10 G55 ;                                        Preparations for the programming coordinate system
N20 G90 X0 Y0 Z300.0 B0 C0 ;                     Movement to the initial position
N30 G01 G43.5 H01 Z40.0 F500.0 ;                 Start of tool center point control
                                                 H01 is a tool length compensation number
N40 G41.6 D01;                                   Start of cutter compensation
                                                 D01 is a tool cutter compensation number
N50 X50.0 Y50.0 Z20.0 I35.355 J35.355 K86.603 ; The Z-axis height on the machining plane is 20.
N60 X-50.0 I-35.355 J35.355 K86.603 ;
N70 X-100.0 Y-100.0 I-35.355 J-35.355 K86.603 ;
N80 X100.0 I35.355 J-35.355 K86.603 ;
N90 X50.0 Y50.0 I35.355 J35.355 K86.603 ;
N100 X0 Y0 Z40.0 K1.0 ;
N110 G40 ;                                       Cutter compensation cancellation
N120 G49 Z300.0 ;                                Tool center point control cancellation
                                                 Movement to the initial position on the Z-axis
N130 M30;
```

By using type 2 as in program 3, the same program can be used with machines with different configurations, regardless of whether the machine configuration is the tool rotation type, table rotation type, or composite type.

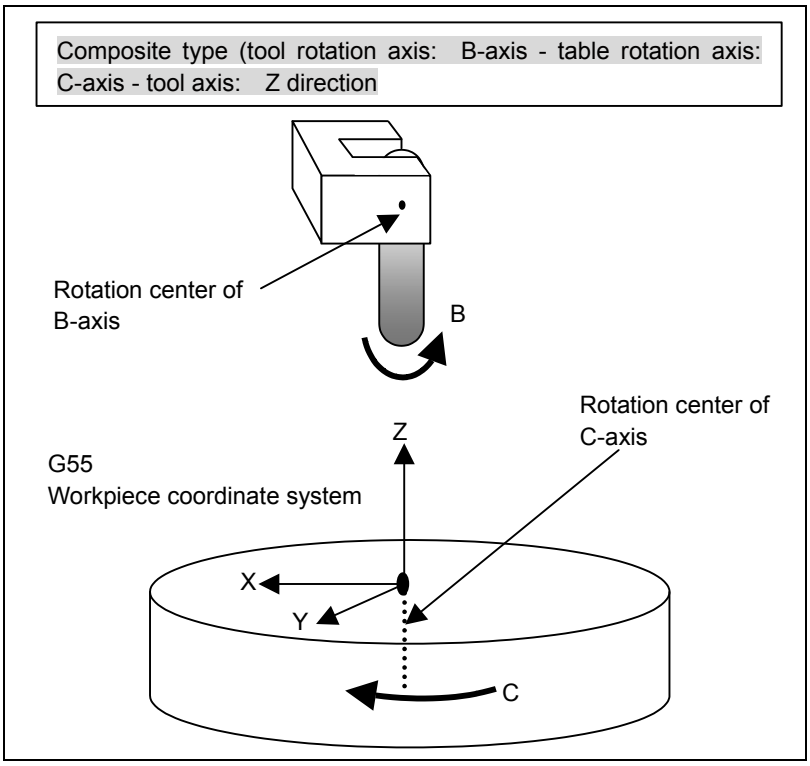


Fig. 22.8.6 (a) Machine configuration for example

Fig. 22.8.6 (b) shows the attitudes of the workpiece (object to be machined) and the tool head (relative to the workpiece (object to be machined)) as viewed in the positive Z direction of the programming coordinate system fixed to the table (table coordinate system).

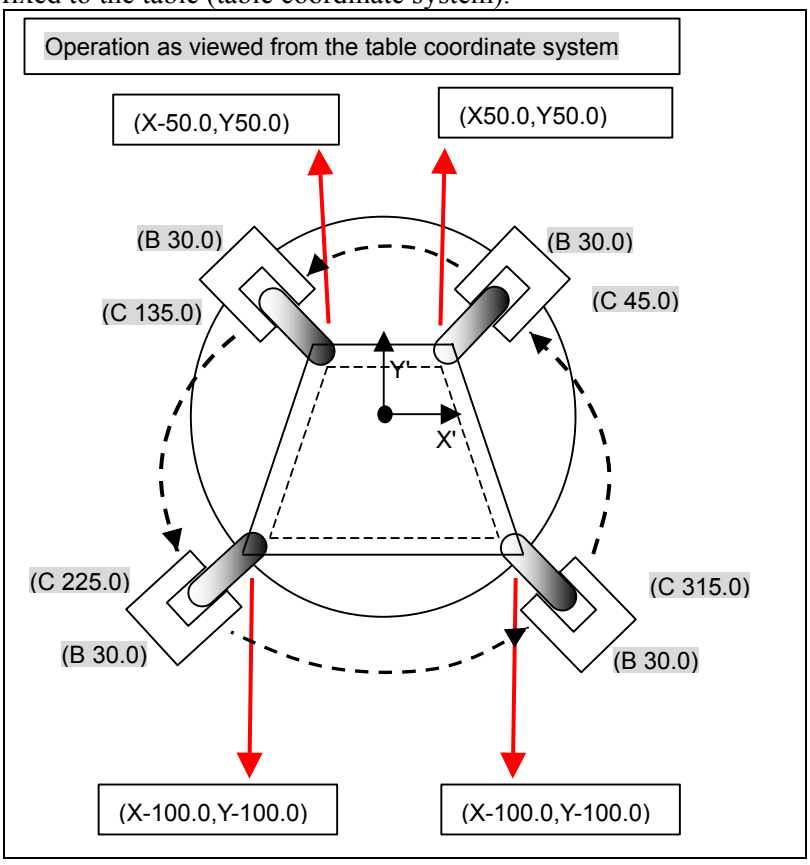


Fig. 22.8.6 (b) Illustration of example

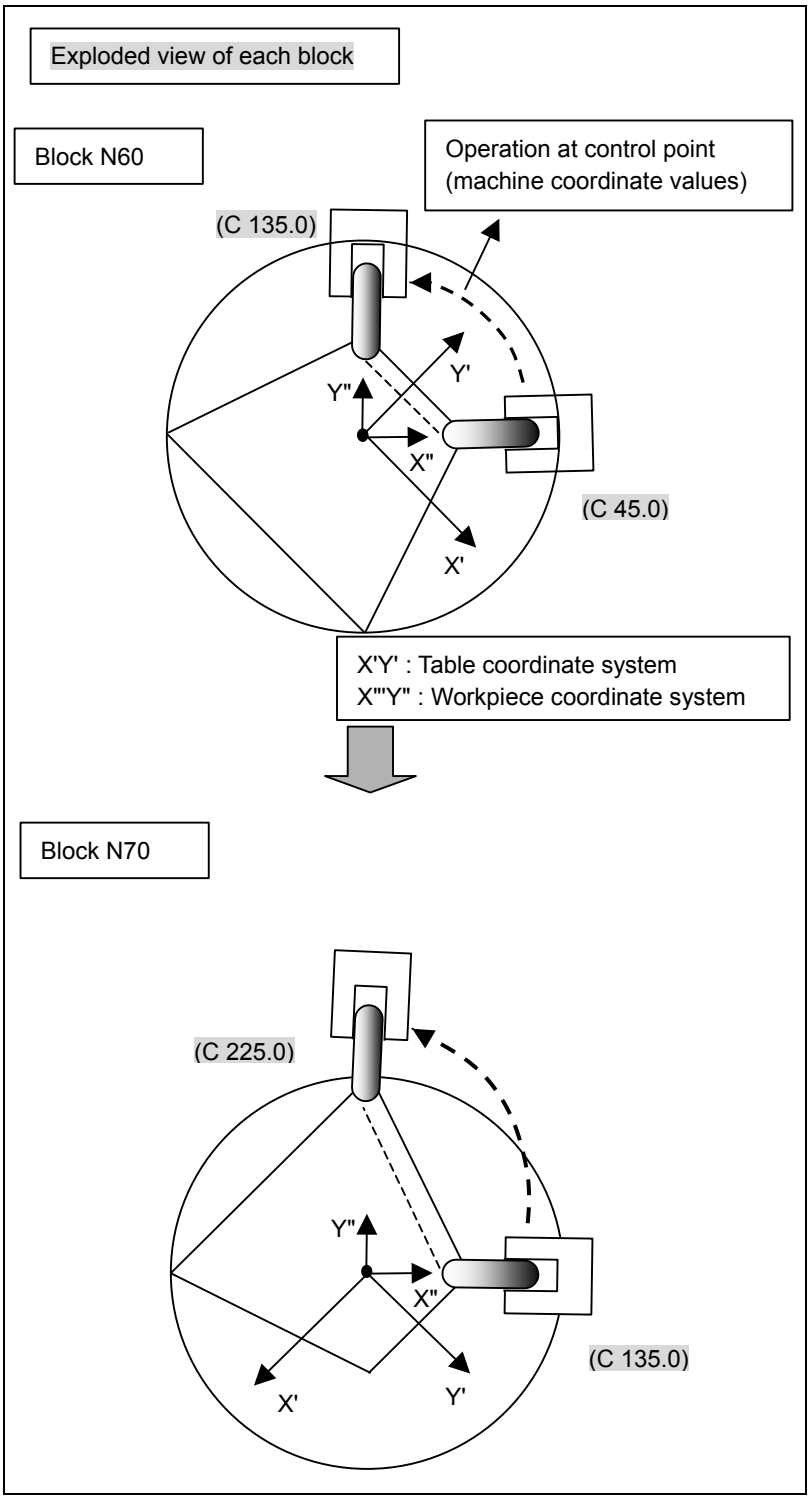


Fig. 22.8.6 (c) Exploded View of Each Block (1)

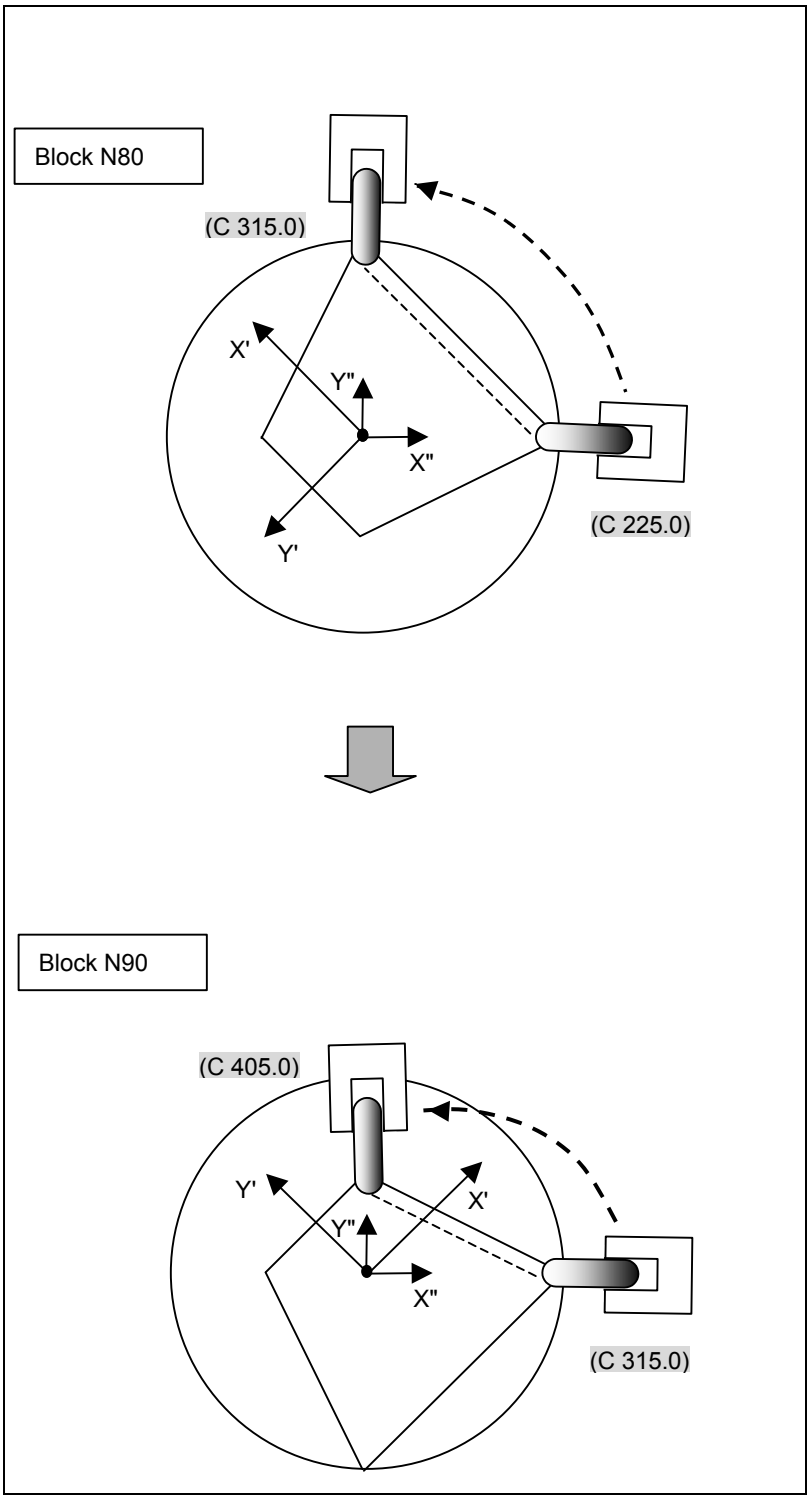


Fig. 22.8.6 (d) Exploded View of Each Block (2)

22.9 EXPANSION OF THE WAY TO SET 5-AXIS MACHINING FUNCTION PARAMETERS

Overview

By setting bit 7 (SPM) of parameter No. 19754 to 1, the parameters of the 5-axis machining functions can be set with reference to the machine coordinate of 0 on the rotation axis. This makes it possible to use the 5-axis machining functions without changing the parameters even if a workpiece offset value is set on the rotation axis.

Explanation

The conventional parameters of the 5-axis machining functions (Nos. 19681 to 19714) are set with reference to the absolute coordinate of 0 on the rotation axis. By setting bit 7 (SPM) of parameter No. 19754 to 1, the machine coordinate of 0 on the rotation axis is used as the reference. This makes it possible to use the 5-axis machining functions without changing the parameters even if a workpiece offset value is set on the rotation axis.

This function is effective to the following functions:

- Tool center point control
- Smooth TCP
- Tool posture control
- 3-dimensional tool compensation
- Cutting point command
- Workpiece setting error compensation
- Tilted working plane indexing

Setting example (tool rotation type)

This function explained below, using a tool rotation type machine that looks like in Fig. 22.9 (a), below, when the machine coordinates are C0B0. (C-axis (master axis): About the Z-axis, B-axis (slave axis): About the Y-axis)

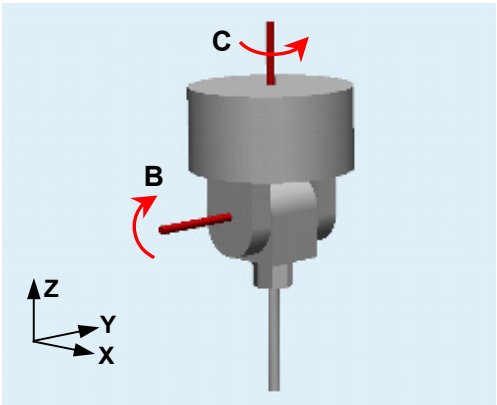


Fig. 22.9 (a) Machine coordinates B0 C0

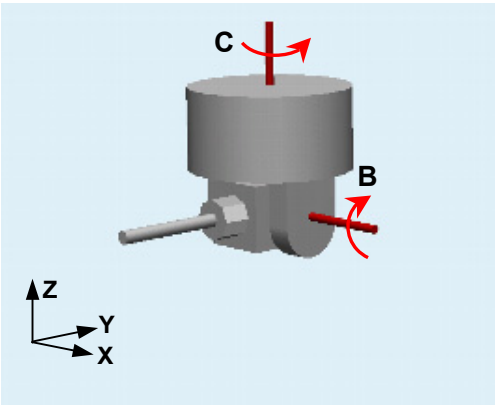


Fig. 22.9 (b) Machine coordinates B90 C90

If the workpiece offset value (C,B) is equal to (0,0), the machine configuration with the absolute coordinates of C0B0 (machine coordinates: C0B0) (Fig. 22.9 (a)) is as below.

- First rotation axis: About the Z-axis
- Second rotation axis: About the Y-axis
- Reference tool axis direction: Z-axis direction
- Crossover offset vector of the tool axis and the second rotation axis: (X_{t2}, Y_{t2}, Z_{t2})
- Crossover offset vector of the second rotation axis and the first rotation axis: (X_{21}, Y_{21}, Z_{21})

If the workpiece offset value (C,B) is changed to (90,90), the machine configuration with the absolute coordinates of C0 B0 (machine coordinates: C90B90) (Fig. 22.9 (b)) changes as below.

- Second rotation axis: About the -X-axis
- Reference tool axis direction: Y-axis direction
- Crossover offset vector of the tool axis and the second rotation axis: Vector resulting from rotating (X_{t2}, Y_{t2}, Z_{t2}) by the workpiece offset value of the first rotation axis and the second rotation axis.
- Crossover offset vector of the second rotation axis and the first rotation axis: Vector resulting from rotating (X_{21}, Y_{21}, Z_{21}) by the workpiece offset value of the first rotation axis.

If bit 7 (SPM) of parameter No. 19754 is 0 (conventional specifications), the parameters related to the 5-axis functions must be changed considering the changes in machine configuration due to the workpiece offset of the rotation axes.
If bit 7 (SPM) of parameter No. 19754 is 1, the parameters need not be changed because they are set with reference to the machine coordinate of 0 ((Fig. 22.9 (a)).

Setting example (table rotation type)

This function is explained below, using a table rotation type machine that looks like in Fig. 22.9 (c), below, when the machine coordinates are B0C0. (B-axis (master axis): About the Y-axis, C-axis (slave axis) About the Z-axis)

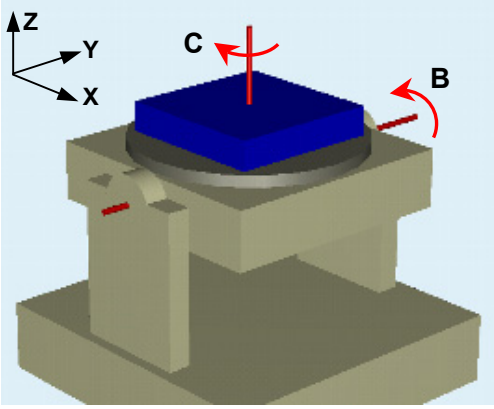


Fig. 22.9 (c) Machine coordinates B0 C0

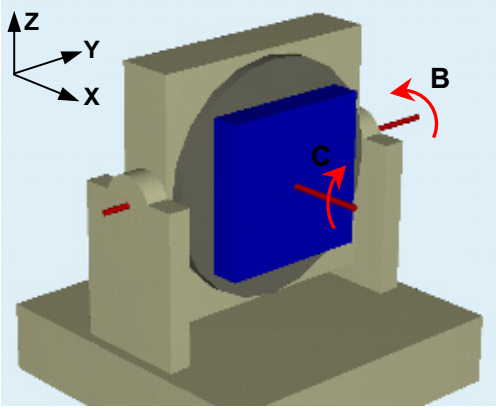


Fig. 22.9 (d) Machine coordinates B-90 C0

If the workpiece offset value (B,C) is equal to (0,0), the machine configuration with the absolute coordinates of B0C0 (machine coordinates: B0C0) (Fig. 22.9 (c)) is as below.

- First rotation axis: About the Y-axis
- Second rotation axis: About the Z-axis
- Crossover offset vector of the table rotation axis: (X,Y,Z)

If the workpiece offset value (B,C) is changed to (-90,0), the machine configuration with the absolute coordinates of B0C0 (machine coordinates: B-90C0) (Fig. 22.9 (d)) changes as below.

- Second rotation axis: About the X-axis
- Crossover offset vector of the table rotation axis: Vector resulting from rotating (X,Y,Z) by the workpiece offset value.

If bit 7 (SPM) of parameter No. 19754 is 0 (conventional specifications), the parameters related to the 5-axis functions must be changed considering the changes in machine configuration due to the workpiece offset of the rotation axes.
If bit 7 (SPM) of parameter No. 19754 is 1, the parameters need not be changed because they are set with reference to the machine coordinate of 0 (Fig. 22.9 (c)).

Setting example (composite type)

This function is explained below, using a composite type machine that looks like in Fig. 22.9 (e), below, when the machine coordinates are B0C0. (B-axis (first rotation axis): About the Y-axis, C-axis (second rotation axis): About the Z-axis)

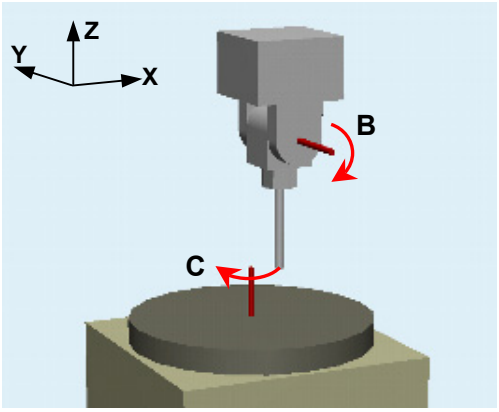


Fig. 22.9 (e) Machine coordinates B0 C0

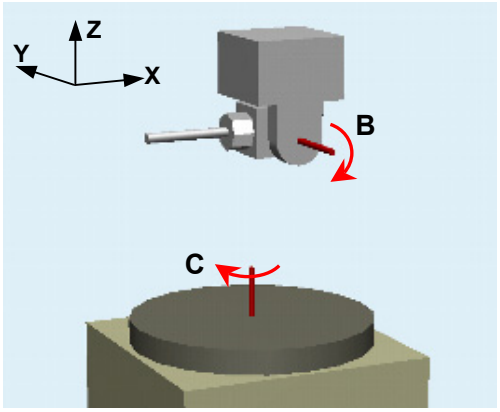


Fig. 22.9 (f) Machine coordinates B90 C0

If the workpiece offset value (B,C) is equal to (0,0), the machine configuration with the absolute coordinates of B0C0 (machine coordinates: B0C0) (Fig. 22.9 (e)) is as below.

- First rotation axis: About the Y-axis
- Second rotation axis: About the Z-axis
- Reference tool axis direction: Z-axis direction
- Crossover offset value of the tool axis and the first rotation axis: (Xt1, Yt1, Zt1)

If the workpiece offset value (B,C) is changed to (90,0), the machine configuration with the absolute coordinates of B0C0 (machine coordinates: B90C0) (Fig. 22.9 (f)) changes as below.

- Reference tool axis direction: X-axis direction
- Crossover offset vector of the tool axis and the first rotation axis: Vector resulting from rotating (Xt1, Yt1, Zt1) by the workpiece offset value of the first rotation axis.

If bit 7 (SPM) of parameter No. 19754 is 0 (conventional specifications), the parameters related to the 5-axis functions must be changed considering the changes in machine configuration due to the workpiece offset of the rotation axes.
If bit 7 (SPM) of parameter No. 19754 is 1, the parameters need not be changed because they are set with reference to the machine coordinate of 0 (Fig. 22.9 (e)).

Limitation

- Inclined rotary axis control

This function cannot be used if the rotary axis is inclined due to inclined rotary axis control. If an attempt is made to use it, alarm PS5459 occurs.

- 3-dimensional manual feed / Thermal growth compensation along tool vector

If this function is used together with 3-dimensional manual feed/thermal growth compensation along tool vector, the operation is the same as that if bit 0 (CAC) of parameter No. 12319 and bit 1 (CAT) of parameter No. 12319 are set to 0. (The coordinate system of the rotation axis used for calculation is the machine coordinate system.)

22.10 MACHINE CONFIGURATION SELECTING FUNCTION

Overview

The Machine Configuration Selecting function allows easy switching machine configuration in the multi-tasking machine system. Ten sets of parameters for machine configuration are saved. One of those can be selected according to the G10.8 L3 command, and the values of selected parameter set are reflected to the actual parameters, and the machine configuration is switched. Therefore, in the machine having Tool head or Rotary table whose position can be changed, parameters for multi-tasking machine configuration can be switched by only one command.
Ten sets of parameters for these machine configurations can be set on machine configuration selecting screen.

22.10.1 Machine Configuration Selecting Screen

Ten sets of parameters for machine configurations are called machine configuration data.
The machine configuration data are independent for each path. The machine configuration data are displayed on the machine configuration selecting screen. The numbers of data displayed in each path are set by parameter No.11266. The number of machine configuration sets that can be used in all paths is up to 10.

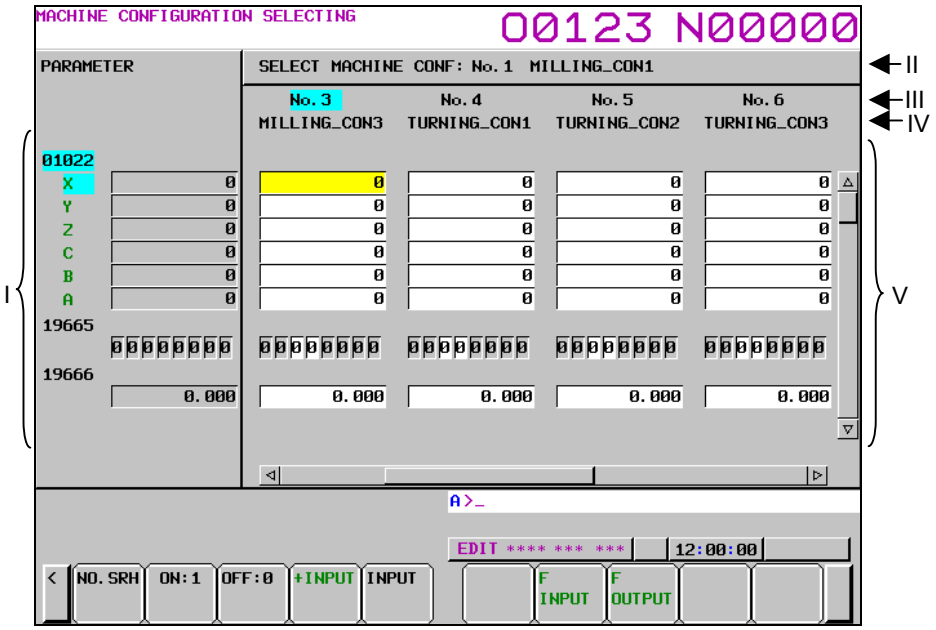


Fig.22.10.1 (a) Machine configuration selecting screen

Screen structure

I Parameter:
Present parameter values in currently displayed path are displayed.

II Active machine configuration set :
The set number and the name of the active machine configuration data are displayed.

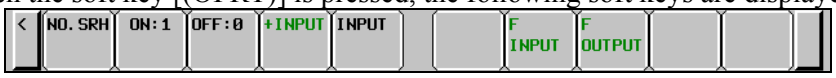
III Set number :
The set numbers of the machine configuration data is displayed.

IV Machine configuration set name :
The names of machine configuration data are displayed.

V Machine configuration data :
The machine configuration data are displayed.

NOTE
The set number and the name last selected by machine configuration selecting command are displayed in II "active machine configuration set". Therefore, when the NC parameter is rewritten or machine configuration data are changed, present NC parameter and machine configuration data might be different.
When a present NC parameter and the parameter of machine configuration data under the selecting are different, the parameter value of machine configuration data is displayed in red character. Moreover, the set number and the name of screen upper "machine configuration under selecting " are displayed in red character.

Additionally, when the soft key [(OPRT)] is pressed, the following soft keys are displayed.



[NO.SRH] : Input the searched data number and press this soft key. The cursor moves to the target data.
[ON:1] : When the soft key is pressed ,'1' is set to the bit type data on cursor position.
[OFF:0] : When the soft key is pressed ,'0' is set to the bit type data on cursor position.
[+INPUT] : After a number is entered, pressing soft key [+INPUT] causes the number to be added.
[INPUT] : After a number is entered, pressing soft key [INPUT] causes the number to be set.
[F INPUT] : Machine configuration data are read from an external device.
[F OUTPUT] : Machine configuration data are output to an external device.

22.10.2 Switching Machine Configuration

Machine configuration data of the set number specified by the instruction of machine configuration selecting command in each path are reflected to the present NC parameters. As a result, machine configuration can be easily switched.
Machine configuration data of the selected set number are reflected to the NC parameters by executing the following instruction by the program.

Format

Machine setting selecting
G10.8 L3 P_;
P_ : Machine configuration set No.1 to 10

Example

Example program for switching machine configuration
N010 G10.8 L3 P7; Switching to No.7 of machine configuration
N020 G54 G90 G00 X0 Y0 Z0 B0 C0;

If specifying G10.3 L3 is necessary, specify it only after canceling the following 5-axis machining mode.
Tool center point control
High-speed smooth TCP
Tool posture control
Cutting point command
Tilted working plane indexing command
3-dimensional cutter compensation
Workpiece setting error compensation
3-dimensional manual feed
Thermal growth compensation along tool vector

Three-dimensional rotary error compensation
Tool axis direction tool length compensation

NOTE
1 Bit 7 (MSF) of parameter No. 11269 decides Enabled (1)/ Disabled (0) of the G10.8L3 command. If G10.8L3 is specified in MSF = 0, the alarm PS0010 "IMPROPER G-CODE" is issued.
2 If G10.8L3 is specified in the following modes, each alarm is issued.
(1) Tool center point control :
PS5421 “ILLEGAL COMMAND IN G43.4/G43.5”
(2) Smooth TCP :
PS5421 “ILLEGAL COMMAND IN G43.4/G43.5”
(3) Tool posture control :
PS5421 “ILLEGAL COMMAND IN G43.4/G43.5”
(4) Cutting point command :
PS5464 “ILLEGAL COMMAND IN G43.8/G43.9”
(5) Tilted working plane indexing :
PS5462 “ILLEGAL COMMAND (G68.2/G69)”
(6) 3-dimensional cutter compensation :
PS5460 “ILLEGAL USE OF 3-DIMENSIONAL CUTTER COMPENSATION”
(7) Workpiece setting error compensation :
PS0437 “ILLEGAL COMMAND IN WSC”

Machine configuration data

The value corresponding to the following parameter is set to machine configuration data.

Table22.10.2 (a) Parameters that can be set

Parameter No.	Description	Data type
1022	Setting of each axis in the basic coordinate system	Byte axis
19665#4	0 : The controlled point is shifted by automatic calculation. 1 : The controlled point is shifted by using parameter No. 19667.	Bit path
19665#5	0 : The controlled point is not shifted. 1 : The controlled point is shifted.	Bit path
19666	Tool holder offset value	Real path
19667	Controlled-point shift vector	Real axis
19680	Mechanical unit type	Byte path
19681	Controlled axis number for the first rotation axis	Byte path
19682	Axis direction of the first rotation axis	Byte path
19683	Inclination angle when the first rotation axis is an inclined axis	Real path
19684	Rotation direction of the first rotation axis	Byte path
19685	Rotation angle when the first rotation axis is a hypothetical axis	Real path
19686	Controlled axis number for the second rotation axis	Byte path
19687	Axis direction of the second rotation axis	Byte path
19688	Inclination angle when the second rotation axis is inclined	Real path
19689	Rotation direction of the second rotation axis	Byte path
19690	Rotation angle when the second rotation axis is a hypothetical axis	Real path
19696#0	0 : The first rotation axis is an ordinary rotation axis. 1 : The first rotation axis is a hypothetical axis.	Bit path
19696#1	0 : The second rotation axis is an ordinary rotation axis. 1 : The second rotation axis is a hypothetical axis.	Bit path
19696#5	0: The table coordinate system is used. 1: The workpiece coordinate system is used.	Bit path
19697	Direction of the reference tool axis	Byte path
19698	Angle when the reference tool axis direction is tilted (reference angle RA)	Real path

Parameter No.	Description	Data type
19699	Angle when the reference tool axis direction is tilted (reference angle RB)	Real path
19700	Rotary table position (X-axis of the basic three axes)	Real path
19701	Rotary table position (Y-axis of the basic three axes)	Real path
19702	Rotary table position (Z-axis of the basic three axes)	Real path
19703	Intersection offset vector between the first and second rotation axes of the table (X-axis of the basic three axes)	Real path
19704	Intersection offset vector between the first and second rotation axes of the table (Y-axis of the basic three axes)	Real path
19705	Intersection offset vector between the first and second rotation axes of the table (Z-axis of the basic three axes)	Real path
19709	Intersection offset vector between the tool axis and tool rotation axis (X-axis of the basic three axes)	Real path
19710	Intersection offset vector between the tool axis and tool rotation axis (Y-axis of the basic three axes)	Real path
19711	Intersection offset vector between the tool axis and tool rotation axis (Z-axis of the basic three axes)	Real path
19712	Intersection offset vector between the second and first rotation axes of the tool (X-axis of the basic three axes)	Real path
19713	Intersection offset vector between the second and first rotation axes of the tool (Y-axis of the basic three axes)	Real path
19714	Intersection offset vector between the second and first rotation axes of the tool (Z-axis of the basic three axes)	Real path
19741	Upper limit of the movement range of the first rotation axis	Real path
19742	Lower limit of the movement range of the first rotation axis	Real path
19743	Upper limit of the movement range of the second rotation axis	Real path
19744	Lower limit of the movement range of the second rotation axis	Real path

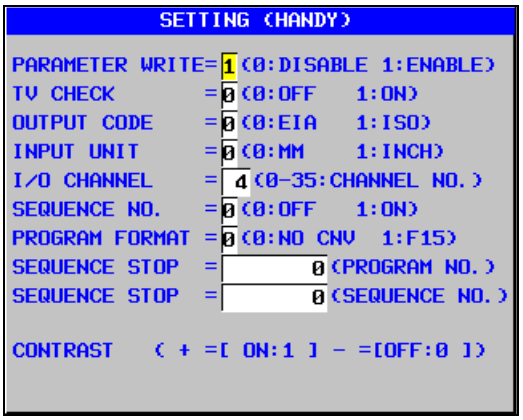
22.10.3 Setting Machine Configuration Data

Machine configuration data can be set by the following 3 methods.
Setting from machine configuration data screen.
Setting by G10.
Input from external device.

The setting method is explained as follows.

Setting from machine configuration data screen (Setting by MDI key)

1 Press the function key .
2 Press the continuous menu key until soft key [SETTING] appears.
Press the soft key [SETTING].
3 Press the MDI switch on the machine operator's panel or enter state emergency stop.
4 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data.

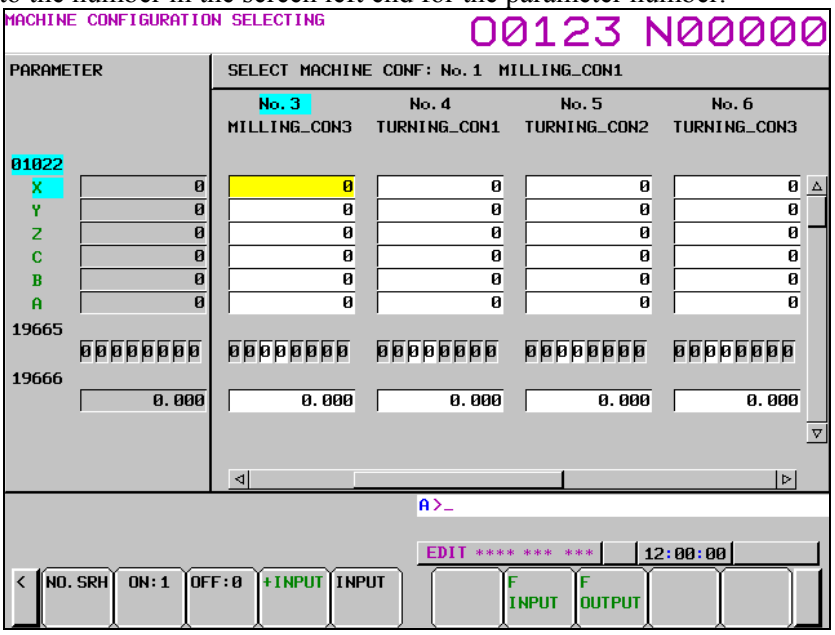


5 Press the soft key [<].

6 Press the continuous menu key ▷ until soft key [MACHINE CONFIG] appears.
Press the soft key [MACHINE CONFIG].



7 Move the cursor to the parameter number of a machine configuration data you want to set by pressing page keys PAGE ↑ and/or PAGE ↓ and cursor keys ↑, ↓, ←, and/or →.
Please refer to the number in the screen left end for the parameter number.



8 Enter the value that is set to the parameter with numeric keys.

9 For direct input of the value set, press the key INPUT or the [INPUT] soft key.
For incremental input of the value set, press the [+INPUT] soft key.

Setting by G10

Machine configuration data and name can be set by specifying G10.

Format

G10 L25 ;	**Machine configuration data entry mode**
I_ P_ N_ R_ ;	**Configuration data : Real path type, Byte path type**
I_ P_ N_ Q_ R_ ;	**Configuration data : Bit path type**
I_ P_ N_ A_ R_ ;	**Configuration data : Byte axis type, Real axis type**
I_ P_ <*********>;**	**Configuration name**
G11 ;	**Cancel machine configuration data entry mode**

Configuration data
I_ : Path number to use machine configuration set for
P_ : Machine configuration set number 1 to 10
N_ : Parameter number to which machine configuration corresponds
R_ : Parameter setting value
Q_ : Bit number
A_ : Axis number
* : Machine configuration name (12 characters or less in ASCII code)

Example

(1) Setting configuration data : byte path type (No.19680)
```
G10 L25 ;
I2 P1 N19680 R21 ;   No.19680 of the 1st set of path 2 = 21 (Mechanical unit type is mixed type)
G11 ;
```

(2) Setting configuration data : bit path type (No.19665#4#5)
```
G10 L25 ;
I2 P2 N19665 Q4 R0 ;   No.19665#4 of the 2nd set of path 2 = 0 (The controlled point is shifted by
                       automatic calculation)
I2 P2 N19665 Q5 R1 ;   No.19665#5 of the 2nd set of path 2 = 1 (The controlled point is shifted)
G11 ;
```

(3) Setting configuration data : real axis type (No.19667)
(When the increment systems for the 1st, 2nd and 3rd axis are IS-B and metric input, respectively)
```
G10 L25 ;
I1 P1 N19667 A1 R4.500 ;    the 1st axis No.19667 of the 1st set of path 1 = 4.500
                            (Controlled-point shift vector 1st axis = 4.500)
I1 P1 N19667 A2 R12.000 ;   the 2nd axis No.19667 of the 1st set of path 1 = 12.000
                            (Controlled-point shift vector 2nd axis = 12.000)
I1 P1 N19667 A3 R0 ;        the 3rd axis No.19667 of the 1st set of path 1 = 0.000
                            (Controlled-point shift vector 3rd axis = 0.000)
G11 ;
```

NOTE
In the value of the real type data R, the unit of the command without decimal point becomes as follows by the parameter DPI (bit 0 of No. 3401).
0: Least input increment is assumed.
1: mm, inch, and degree units are assumed.

(4) Setting configuration name :
```
G10 L25 ;
```

```
I2 P3 <MILLING_CON3>;     The name of configuration the 3rd set for the 2nd path =
                          “MILLING_CON3”
I2 P4 <TURNING_CON1>;     The name of configuration the 4th set for the 2nd path =
                          “TURNING_CON1”
G11 ;
```

22.10.4 Inputting and Outputting Machine Configuration Data

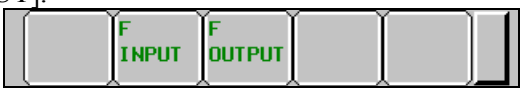
Machine configuration data can be input and output by the operation on the machine configuration data screen. The operating procedure is as follows.

Inputting machine configuration data

1. Make sure the input device is ready for reading.
2. Press the function key .
3. Press the continuous menu key  until soft key [MACHINE CONFIG] appears.
4. Press the soft key [MACHINE CONFIG].



5. Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
6. Press the soft key [OPRT].
7. Press the soft key [F INPUT].



8. Type the name of the file that you want to input.
   If the file name is omitted, default input file name “MACHCON.TXT” is assumed.
9. Press the soft key [EXEC].
   This starts reading the machine configuration data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
   To cancel the input of the program, press the soft key [CAN].

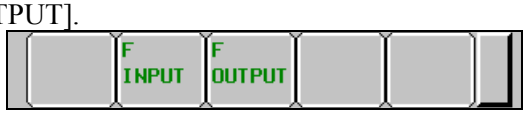


Outputting machine configuration data

1. Make sure the input device is ready for output.
2. Press the function key .
3. Press the continuous menu key  until soft key [MACHINE CONFIG] appears.
4. Press the soft key [MACHINE CONFIG].



5. Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
6. Press the soft key [OPRT].
7. Press the soft key [F OUTPUT].



8. Type the name of the file that you want to output.
   If the file name is omitted, default input file name “MACHCON.TXT” is assumed.

9. Press the soft key [EXEC].
   This starts outputting the machine configuration data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
   To cancel the input of the program, press the soft key [CAN].



Format

The basis is the same as the specifying G10.
The value of the real type data is input and output with decimal point.

23 MUITI-PATH CONTROL FUNCTION

Chapter 23, "MUITI-PATH CONTROL FUNCTION", consists of the following sections:

23.1 OVERVIEW 911
23.2 WAITING FUNCTION FOR PATHS.....912
23.3 COMMON MEMORY BETWEEN EACH PATH 917
23.4 SPINDLE CONTROL BETWEEN EACH PATH 918
23.5 SYNCHRONOUS/COMPOSITE/SUPERIMPOSED CONTROL 919

23.1 OVERVIEW

The multi-path control function is designed to enable 10 independent simultaneous machining with up to 10 paths (10-path control). This function is applicable to lathes and automatic lathes which perform cutting simultaneously with multiple tool posts, combined machine tools which perform turning and milling simultaneously with multiple paths, and machines which require additional control paths such as a loader control path.

For multi-path simultaneous machining, each machining program is stored in a folder in program memory for each path. When automatic operation is to be performed, each path is activated after selecting a program for machining with path 1 and programs for machining with paths 2 to 10 from the programs stored in the respective folders in program memory. Then the programs selected for the tool posts are executed independently at the same time. When tool post 1 and tool post 2 need to wait for each other during machining, the waiting function is available.
Other available functions specific to multi-path control include interference check for each path, balance cut, synchronous control, composite control, spindle control between each path, and common memory between each path.

Just one LCD/MDI is provided for the all paths. Before operation and display on the LCD/MDI, the path selection signal is used to switch between the paths.

Example) For a system with four paths

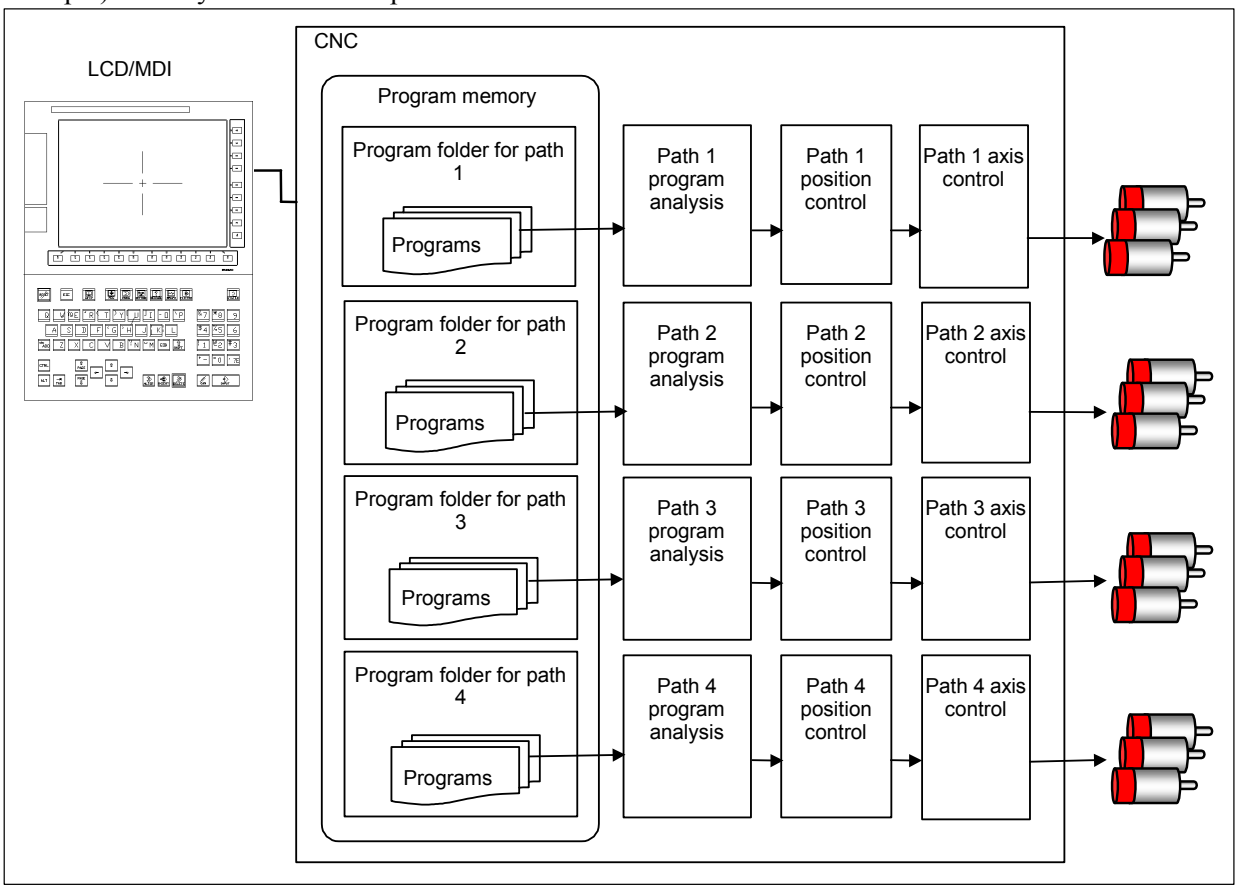


Fig. 23.1 (a)

23.2 WAITING FUNCTION FOR PATHS

Overview

Control based on M codes is used to cause one path to wait for the other during machining. When an M code for waiting is specified in a block for one path during automatic operation, the other path waits for the same M code to be specified before staring the execution of the next block.

A range of M codes used as M codes for waiting is to be set in the parameters Nos. 8110 and 8111 beforehand. Waiting can be ignored using a signal.

Format

M m (Pp) ;

m: Number of an M code for waiting

p: (1) In the binary value specification mode, specify the sum of the binary values corresponding to the numbers of paths which are to wait for one another.

(2) In the path number specification mode, specify the numbers of all paths that are to wait for one another in combination.

Explanation

The following two methods for specifying paths which are to wait for one another at address P in the same block in which an M code for waiting is specified are available and can be selected using bit 1 (MWP) of parameter No. 8103. One method is to specify the paths with the sum of their corresponding binary values (waiting set for three or more paths specified with binary values). The other is to specify them with their path numbers in combination (waiting specified with path numbers).

When address P is not specified, paths 1 and 2 wait for each other (waiting set for two paths). Always specify address P in a single block.

- Waiting specified with binary values

When bit 1 (MWP) of parameter No. 8103 is set to 0, the value specified at address P is assumed to be obtained using binary values. The following table lists the path numbers and corresponding binary values.

Table 23.2 (a)

Path number	Binary value (decimal number)	Path number	Binary value (decimal number)
1	1	6	32
2	2	7	64
3	4	8	128
4	8	9	256
5	16	10	512

The bit position of each path in binary representation is shown below.

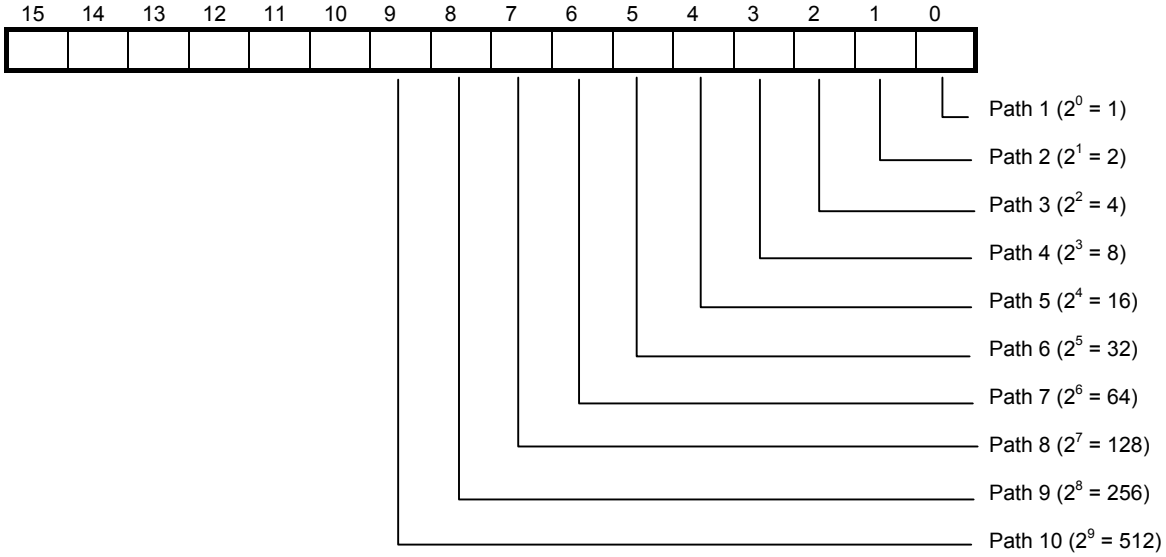


Fig. 23.2 (a)

To make all of paths 1, 2, and 3 wait for one another, the P value is obtained as follows:

Binary value of path 1	1 (0000 0000 0000 0001)
Binary value of path 2	2 (0000 0000 0000 0010)
Binary value of path 3	4 (0000 0000 0000 0100)
Sum	7 (0000 0000 0000 0111)

All of the three paths can be made to wait for one another by specifying P7 together with an M code for waiting.

To make all of paths 1, 3, 5, 7, and 9 wait for one another, the P value is obtained as follows:

Binary value of path 1	1 (0000 0000 0000 0001)
Binary value of path 3	4 (0000 0000 0000 0100)
Binary value of path 5	16 (0000 0000 0001 0000)
Binary value of path 7	64 (0000 0000 0100 0000)
Binary value of path 9	256 (0000 0001 0000 0000)
Sum	341 (0000 0001 0101 0101)

All of the five paths can be made to wait for one another by specifying P341 together with an M code for waiting.

- Waiting specified with a combination of path numbers

When bit 1 (MWP) of parameter No. 8103 is set to 1, the value specified at address P is assumed to be a combination of path numbers. The following table lists the path numbers and corresponding values.

Table 23.2 (b)

Path number	Value (decimal number)	Path number	Value (decimal number)
1	1	6	6
2	2	7	7
3	3	8	8
4	4	9	9
5	5	10	0

To make all of paths 1, 2, and 3 wait for one another, the P value is a number consisting of 1, 2, and 3.
Example) 123
There are no restrictions on the order in which the numeric characters are specified, and the following six possible values can be specified:

P123, P132, P213, P231, P312, P321

Path numbers specified in combination in different orders for different paths are effective as long as the numbers of the relevant paths are specified.
Example)

The following are treated as the same P value and these paths can be made to wait for one another:
M200P123 for path 1, M200P231 for path 2, and M200P321 for path 3

To make all of paths 1, 3, 5, 7, and 9 wait for one another, the P value is a number consisting of 1, 3, 5, 7, and 9.
Example) P13579

- Waiting for path 10

To make path 10 and another path wait for each other, specify a value of 0 for the combination.
If a number begins with 0, 0 cannot be recognized. Specify 0 in the second or subsequent digit from the left.
Incorrect example) P013579
Correct example) P103579
In the incorrect example, the P value is assumed to be the same as P13579. Path 10 cannot be made to wait for other paths and other paths cannot be made to wait for path 10.

Example

- When the value specified at P is obtained using binary values

Assume that the waiting ignore signal for path 2 (G1063.7 for a system with three or more paths) is set to 1 and M101 to M103 (parameter No. 8110 = 101 and parameter No. 8111 = 103) are set as M codes for waiting. In this case, programs O100, O200, and O300 for individual paths are executed as follows:

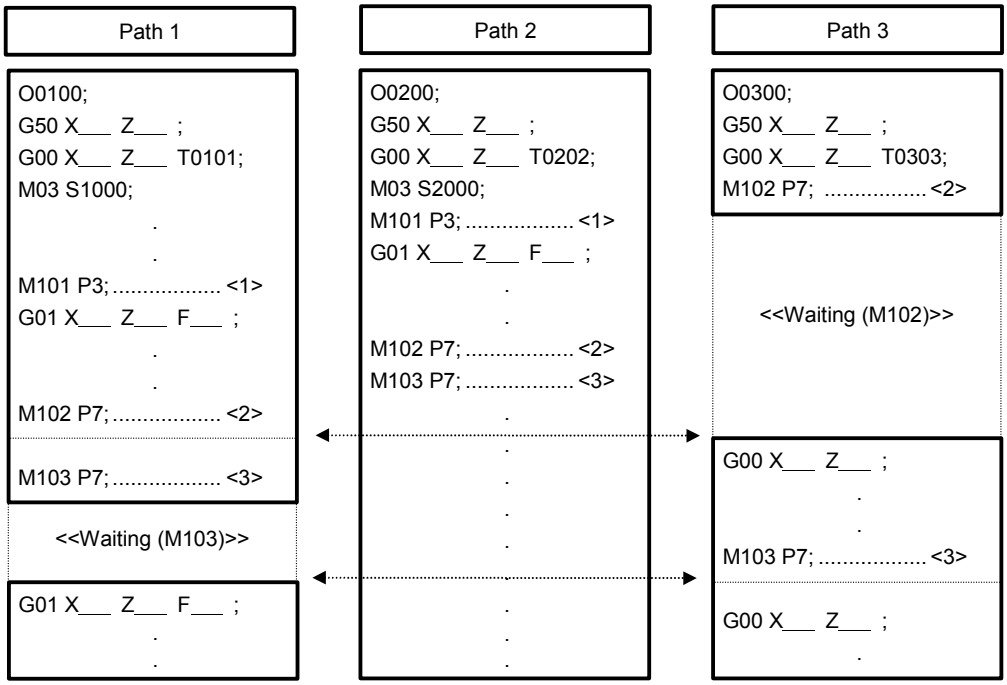


Fig. 23.2 (b)

<1> M101 P3; (making paths 1 and 2 wait for each other)
When the waiting ignore signal for path 2 is set to 0, paths 1 and 2 wait for each other. Because the signal is set to 1, however, paths 1 and 2 ignore the M code for waiting and immediately execute the next block.

<2> M102 P7; (making paths 1, 2, and 3 wait for one another)
In this example, path 3 waits for processing on paths 1 and 2 to terminate. Because the waiting ignore signal for path 2 is set to 1, however, path 3 assumes that waiting is completed when processing on path 1 terminates, and executes the next block.

<3> M103 P7; (making paths 1, 2, and 3 wait for one another)
In this example, paths 1 and 2 wait for processing on path 3 to terminate. Because the waiting ignore signal for path 2 is set to 1, however, path 2 does not wait for processing on path 3 to terminate and executes the next block, but path 1 waits for path 3.

- When the value specified at P is obtained using path numbers in combination

Assume that the waiting ignore signal for path 2 (G1063.7 for a system with three or more paths) is set to 1 and M101 to M103 (parameter No. 8110 = 101 and parameter No. 8111 = 103) are set as M codes for waiting. In this case, programs O100, O200, and O300 for individual paths are executed as follows:

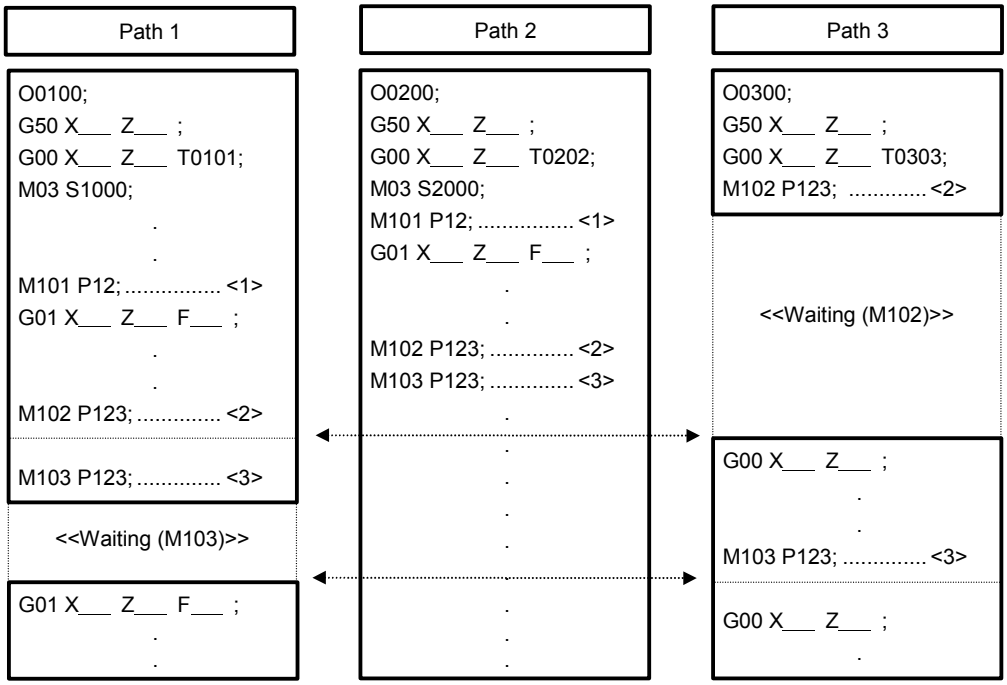


Fig. 23.2 (c)

<1> M101 P12; (making paths 1 and 2 wait for each other)
When the waiting ignore signal for path 2 is set to 0, paths 1 and 2 wait for each other. Because the signal is set to 1, however, paths 1 and 2 ignore the M code for waiting and immediately execute the next block.

<2> M102 P123; (making paths 1, 2, and 3 wait for one another)
In this example, path 3 waits for processing on paths 1 and 2 to terminate. Because the waiting ignore signal for path 2 is set to 1, however, path 3 assumes that waiting is completed when processing on path 1 terminates, and executes the next block.

<3> M103 P123; (making paths 1, 2, and 3 wait for one another)
In this example, paths 1 and 2 wait for processing on path 3 to terminate. Because the waiting ignore signal for path 2 is set to 1, however, path 2 does not wait for processing on path 3 to terminate and executes the next block, but path 1 waits for path 3.

⚠ CAUTION

1 An M code for waiting must always be specified in a single block.
2 Unlike other M codes, the M code for waiting is not output to the PMC.
3 If the operation of a single path is required, the M code for waiting need not be deleted. By using the signal to specify that waiting be ignored (NOWT for the system with two paths, NMWT for the system with three or more paths), the M code for waiting in a machining program can be ignored. For details, refer to the manual supplied by the machine tool builder.
4 If using a waiting M code in block multiple-M code command mode, be sure to specify it as the first M code.

23.3 COMMON MEMORY BETWEEN EACH PATH

Overview

In a multi-path system, this function enables data within the specified range to be accessed as data common to all paths. The data includes tool compensation memory and custom macro common variables.

Explanation

The path common memory function enables the following operations.

- Tool compensation memory

Part or all of tool compensation memory for individual paths can be used as common data by setting parameter No. 5029.

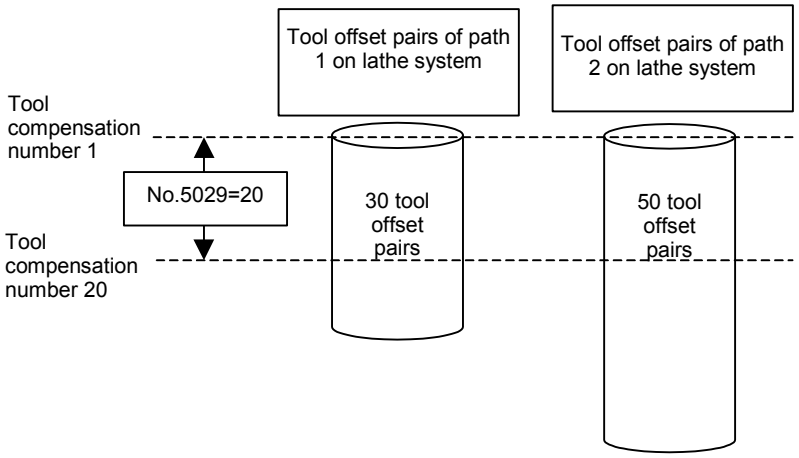


Fig. 23.3 (a)

NOTE
1 When a combined system including a machining center system and lathe system is used, data is made common according to the same path control type.
2 The same unit for tool compensation (bits 0 to 3 of parameter No. 5042) must be set for each of the machining center and lathe systems.
3 Set a value less than the number of tool compensation values for each path for parameter No. 5029.
4 If the value set for parameter No. 5029 is greater than the number of tool compensation values for each path, the minimum number of tool compensation values for individual paths is assumed.
5 For details, refer to the relevant manual of the machine tool builder.

- Custom macro common variables

All or part of custom macro common variables #100 to #149 (, #199, or #499) and #500 to #599 (or #999) can be used as common data by setting parameters No. 6036 (#100 to #149 (, #199, or #499)) and 6037 (#500 to #599 (or #999)).

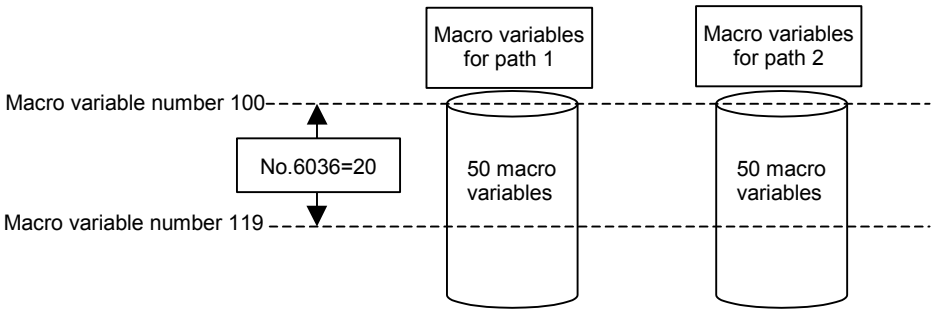


Fig. 23.3 (b)

NOTE
1 If the value of parameter No. 6036 or 6037 exceeds the maximum number of macro common variables, the maximum number of macro common variables is assumed.
2 Common variables #150 to #199, #150 to #499, and #600 to #999 are optional functions.

23.4 SPINDLE CONTROL BETWEEN EACH PATH

Overview

This function allows a workpiece attached to one spindle to be machined simultaneously with two tool posts and each of two workpieces attached to each of two spindles to be machined simultaneously with each of two tool posts.

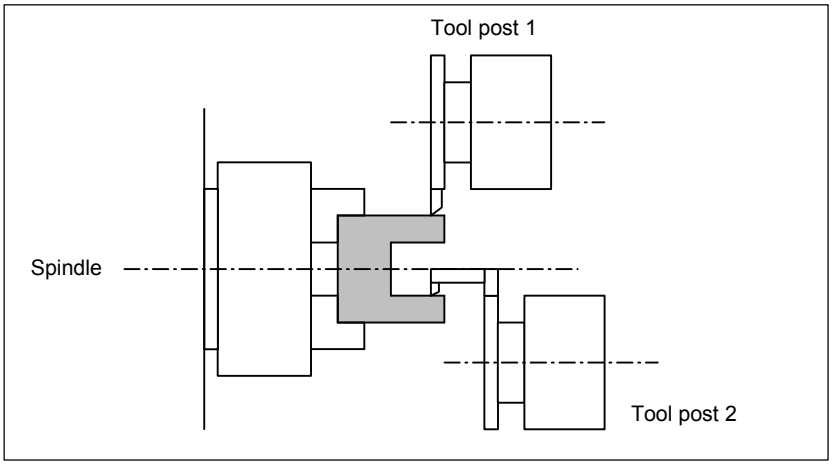


Fig. 23.4 (a) Application to a lathe with one spindle and two tool posts

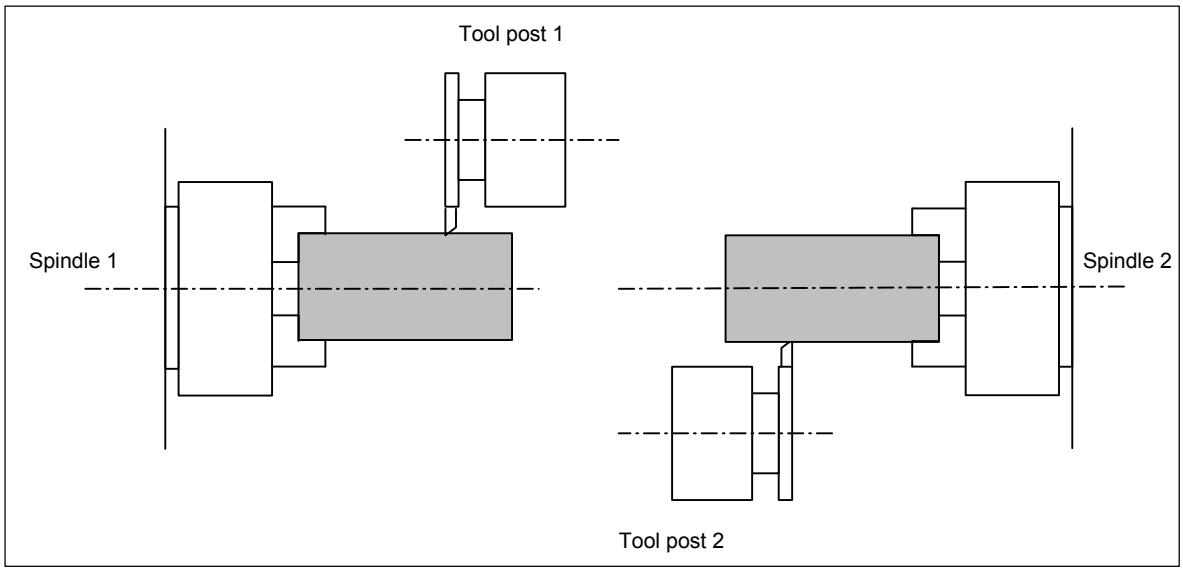


Fig. 23.4 (b) Application to a lathe with two spindles and two tool posts

The spindle belonging to each path can generally be controlled by programmed commands for the path. With path spindle command selection signals, programmed commands for any path can control the spindle belonging to any path.

NOTE
For the method of spindle command selection, refer to the relevant manual of the machine tool builder.

23.5 SYNCHRONOUS/COMPOSITE/SUPERIMPOSED CONTROL

Overview

In multi-path control, the synchronous control function, composite control function, and superimposed control function enable synchronous control, composite control, and superimposed control in a single path system or between multi-path systems.

Explanation

- Synchronous control

- Synchronizes movement along an axis of one system with that along an axis of the other path.
  Example)
  Synchronizing movement along the Z1 (master) and Z2 (slave) axes (in the case of turning)

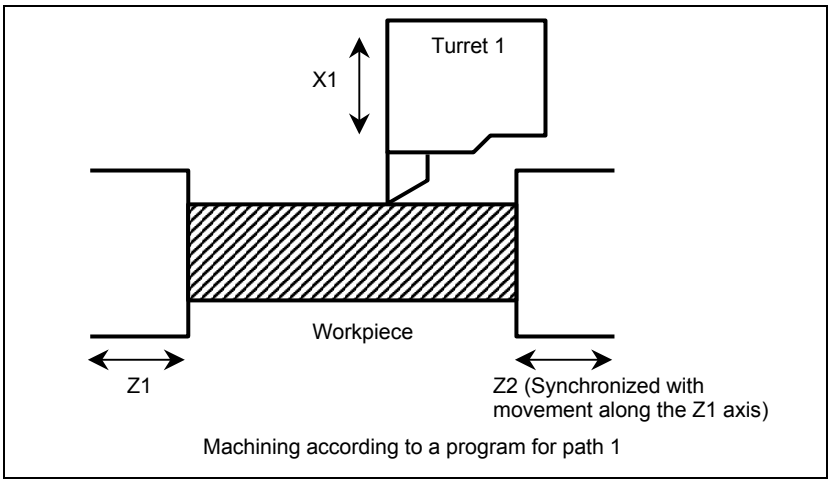


Fig. 23.5 (a)

- Synchronizes movement along an axis of one path with that along another axis of the same path.
  Example)
  Synchronizing movement along the Z1 (master) and B1 (slave) axes (in the case of turning)

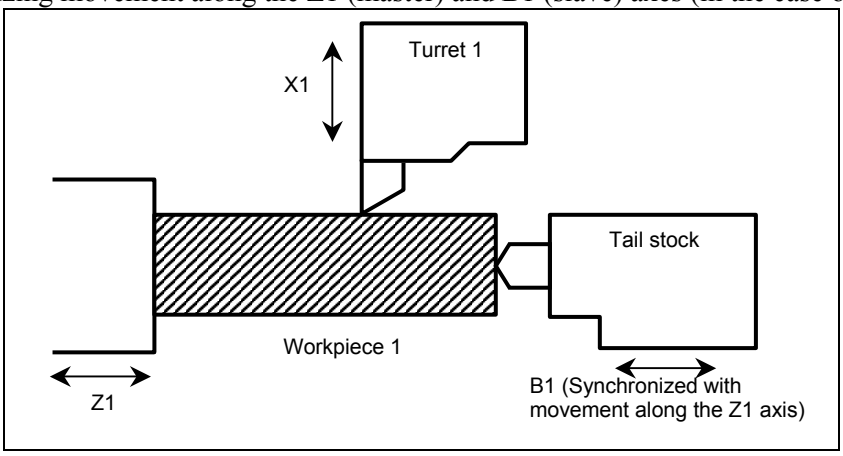


Fig. 23.5 (b)

- Composite control

- Exchanges the move commands for different axes of different path.
  Example)
  Exchanging the commands for the X1 and X2 axes (in the case of turning)
  → Upon the execution of a command programmed for path 1, movement is performed along the X2 and Z1 axes.
  Upon the execution of a command programmed for path 2, movement is performed along the X1 and Z2 axes.

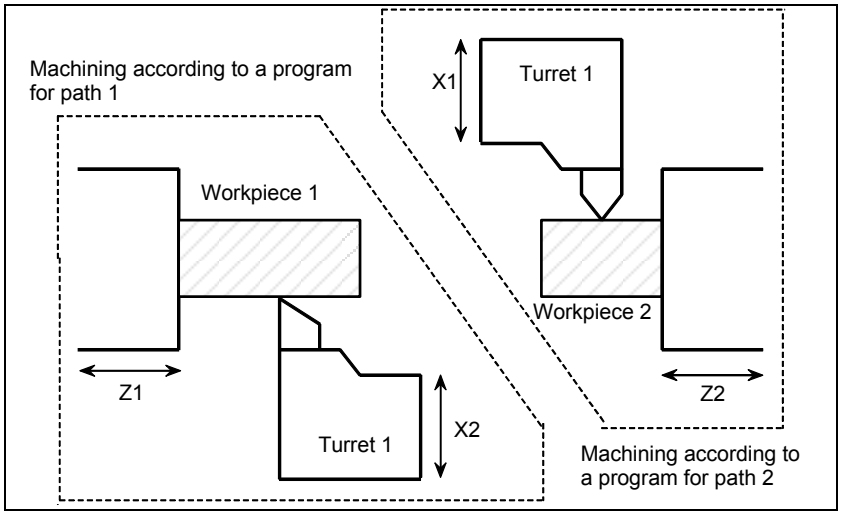


Fig. 23.5 (c)

- Superimposed control

- Provides a move command of an axis for a different axis in another path.
  Example)
  Providing the Z2 (slave) axis with a move command specified for the Z1 (master) axis (in the case of turning)

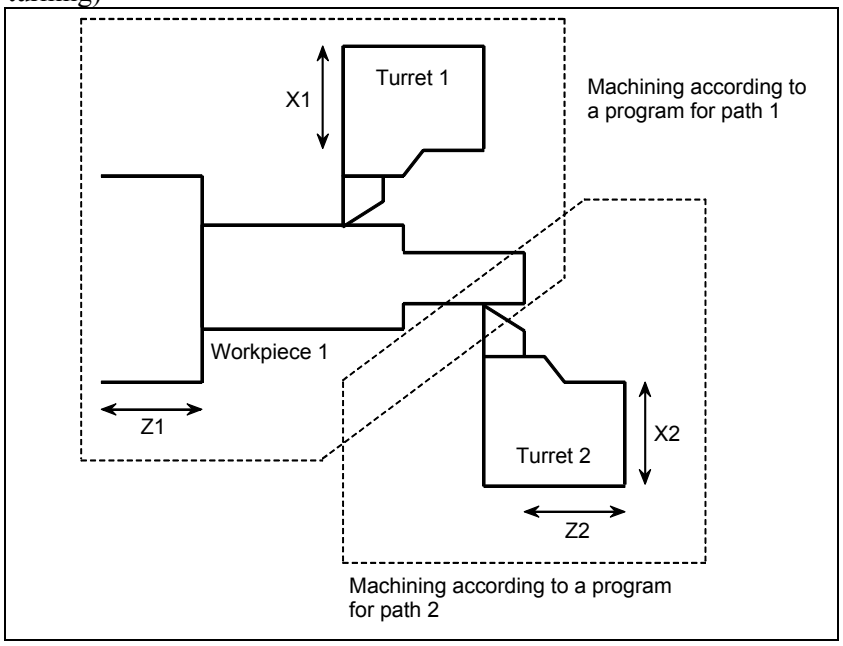


Fig. 23.5 (d)

NOTE
The method used to specify synchronous, composite, or superimposed control varies with the machine tool builder. For details, refer to the manual supplied by the machine tool builder.

III. OPERATION

1 GENERAL

Chapter 1, "GENERAL", consists of the following sections:

1.1 MANUAL OPERATION....925
1.2 TOOL MOVEMENT BY PROGRAMING - AUTOMATIC OPERATION....926
1.3 AUTOMATIC OPERATION....927
1.4 TESTING A PROGRAM....928
1.5 EDITING A PROGRAM....930
1.6 DISPLAYING AND SETTING DATA....930
1.7 DISPLAY....933

1.1 MANUAL OPERATION

Explanation

- Manual reference position return

The CNC machine tool has a position used to determine the machine position.
This position is called the reference position, where the tool is replaced or the coordinate are set. Ordinarily, after the power is turned on, the tool is moved to the reference position.
Manual reference position return is to move the tool to the reference position using switches and pushbuttons located on the operator's panel. (See Section, “MANUAL REFERENCE POSITION RETURN”.)

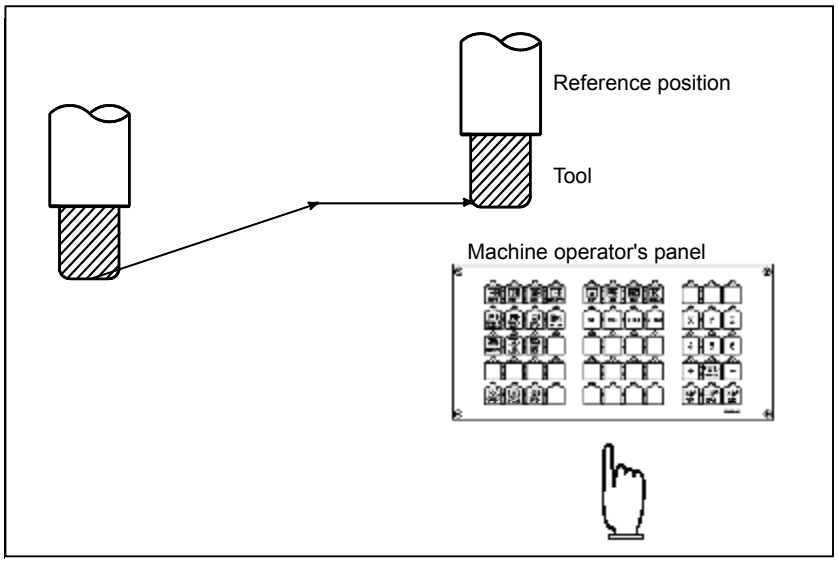


Fig. 1.1 (a) Manual reference position return

The tool can be moved to the reference position also with program commands.
This operation is called automatic reference position return. (See Chapter, “REFERENCE POSITION”.)

- The tool movement by manual operation

Using machine operator's panel switches, pushbuttons, or the manual handle, the tool can be moved along each axis.

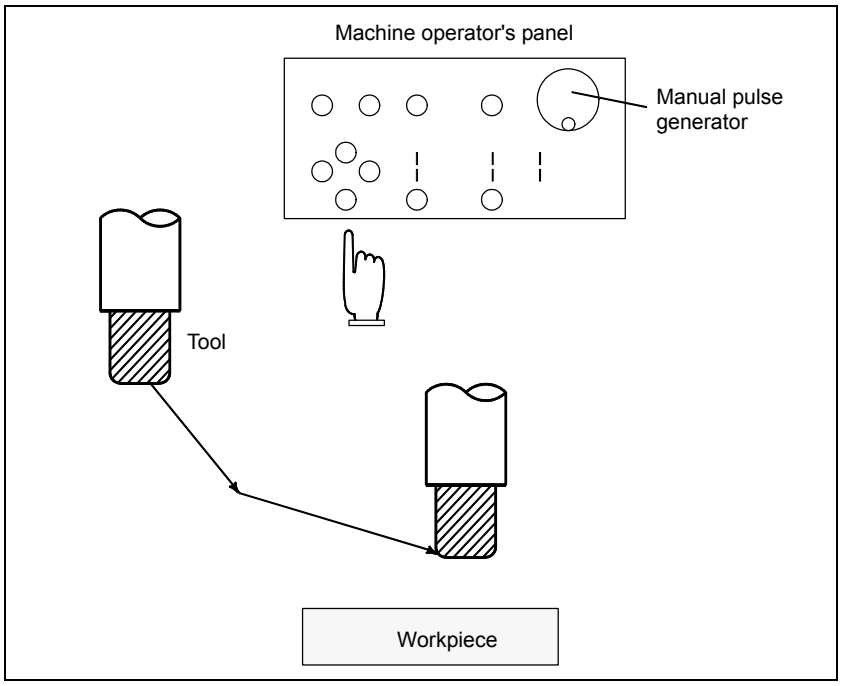


Fig. 1.1 (b) The tool movement by manual operation

The tool can be moved in the following ways:

(i) Jog feed (See Section, "JOG FEED (JOG)".)
The tool moves continuously while a pushbutton remains pressed.

(ii) Incremental feed (See Section, "INCREMENTAL FEED".)
The tool moves by the predetermined distance each time a button is pressed.

(iii) Manual handle feed (See Section, "MANUAL HANDLE FEED".)
By rotating the manual handle, the tool moves by the distance corresponding to the degree of handle rotation.

1.2 TOOL MOVEMENT BY PROGRAMING - AUTOMATIC OPERATION

Automatic operation is to operate the machine according to the created program. It includes memory, MDI and DNC operations. (See Chapter, "AUTOMATIC OPERATION".)

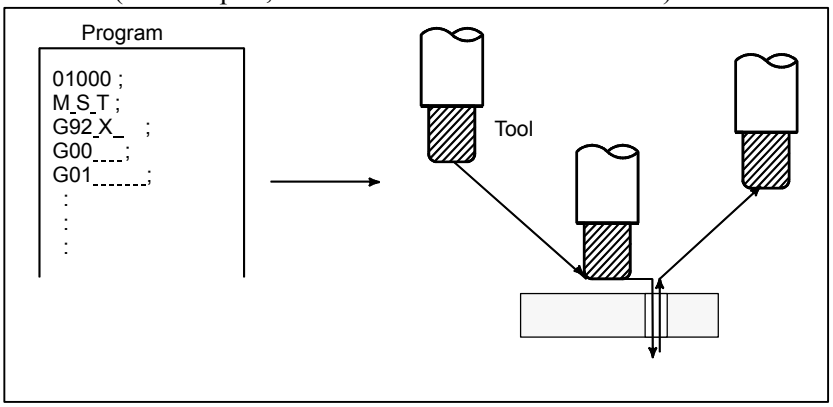


Fig. 1.2 (a) Tool Movement by programming

Explanation

- Memory operation

After the program is once registered in memory of CNC, the machine can be run according to the program instructions. This operation is called memory operation.

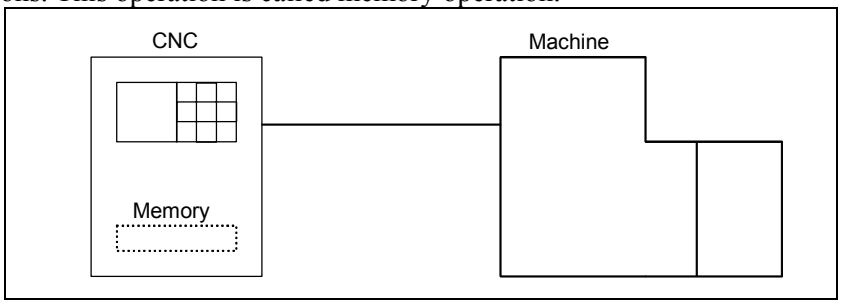


Fig. 1.2 (b) Memory operation

- MDI operation

After the program is entered, as a command group, from the MDI unit, the machine can be run according to the program. This operation is called MDI operation.

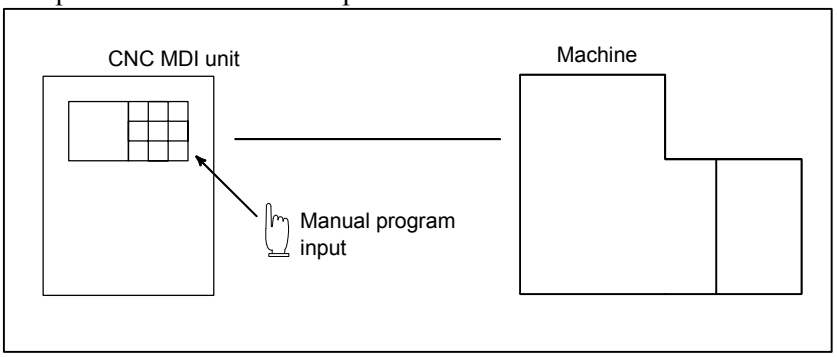


Fig. 1.2 (c) MDI operation

- DNC operation

In this mode of operation, the program is not registered in the CNC memory. It is read from the external input/output devices instead. This is called DNC operation.

1.3 AUTOMATIC OPERATION

Explanation

- Program selection

Select the program used for the workpiece. Ordinarily, one program is prepared for one workpiece. If two or more programs are in memory, select the program to be used, by searching the program number. (See Section, “PROGRAM SEARCH”.)

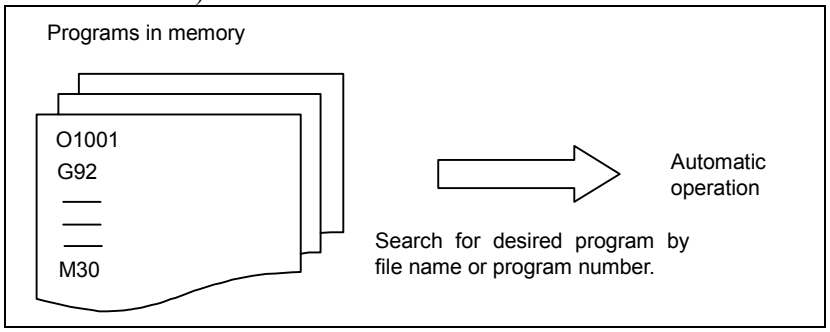


Fig. 1.3 (a) Program selection for automatic operation

- Start and stop

Pressing the cycle start pushbutton causes automatic operation to start. By pressing the feed hold or reset pushbutton, automatic operation pauses or stops. By specifying the program stop or program termination command in the program, the running will stop during automatic operation. When one process machining is completed, automatic operation stops. (See Chapter, “AUTOMATIC OPERATION”.)

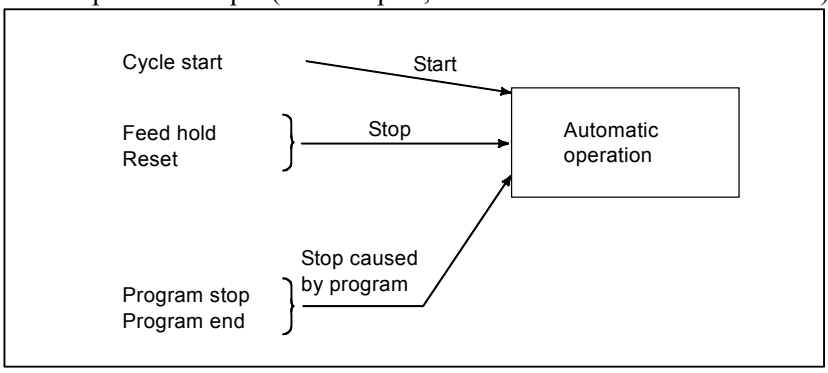


Fig. 1.3 (b) Start and stop for automatic operation

- Handle interruption

While automatic operation is being executed, tool movement can overlap automatic operation by rotating the manual handle. (See Section, “MANUAL HANDLE INTERRUPTION”.)

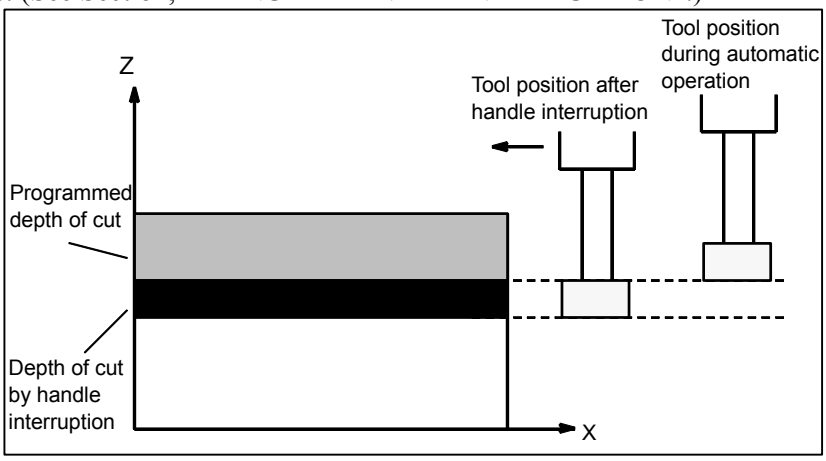


Fig. 1.3 (c) Handle interruption for automatic operation

1.4 TESTING A PROGRAM

Before machining is started, the automatic running check can be executed.
It checks whether the created program can operate the machine as desired.
This check can be accomplished by running the machine actually or viewing the position display change (without running the machine) (See Chapter, “TEST OPERATION”.).

1.4.1 Check by Running the Machine

Explanation

- Dry run

Remove the workpiece, check only movement of the tool. Select the tool movement rate using the dial on the operator's panel. (See Section , “DRY RUN”.)

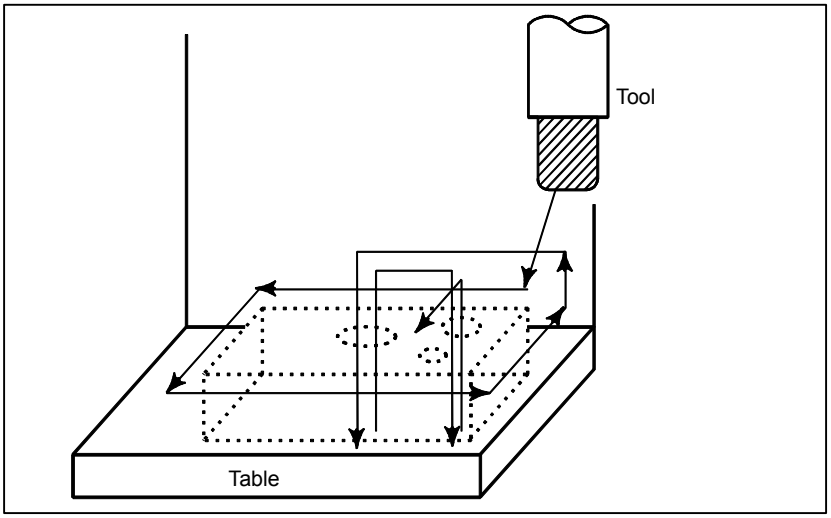


Fig. 1.4.1 (a) Dry run

- Feedrate override

Check the program by changing the feedrate specified in the program. (See Section, "FEEDRATE OVERRIDE".)

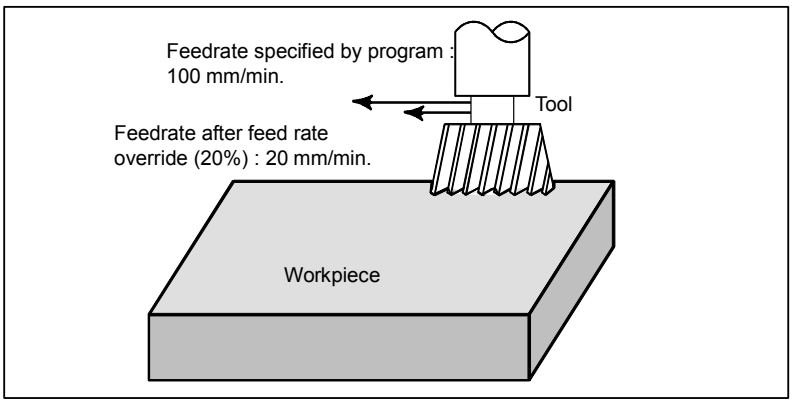


Fig. 1.4.1 (b) Feedrate override

- Single block

When the cycle start pushbutton is pressed, the tool executes one operation then stops. By pressing the cycle start again, the tool executes the next operation then stops. The program is checked in this manner. (See Section, "SINGLE BLOCK".)

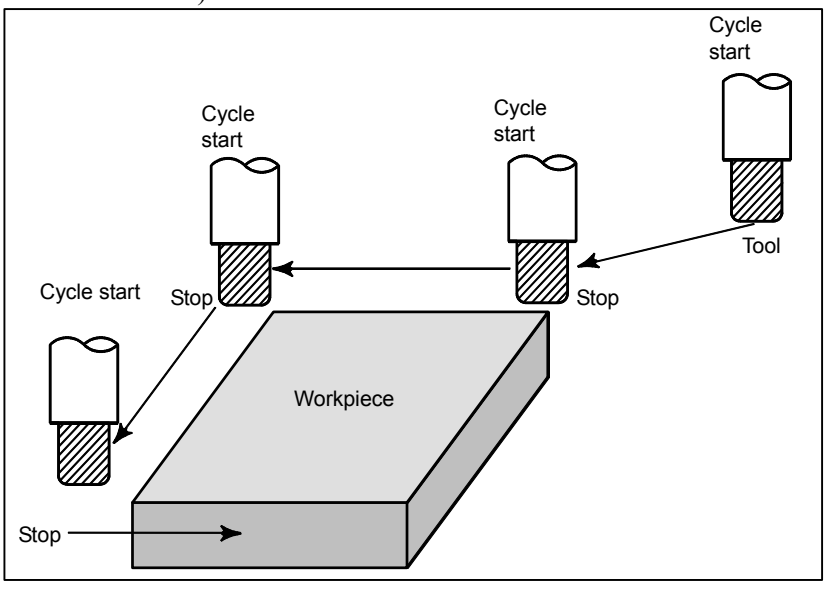


Fig. 1.4.1 (c) Single Block

1.4.2 How to View the Position Display Change without Running the Machine

Explanation

- **Machine Lock**

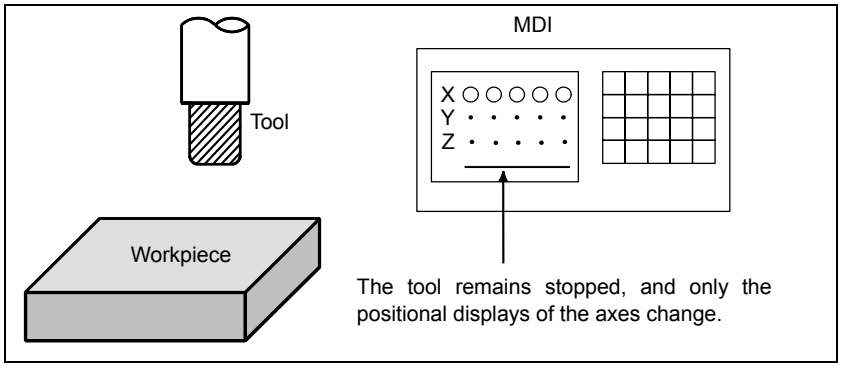


Fig. 1.4.2 (a) Machine Lock

- **Auxiliary function lock**

When automatic running is placed into the auxiliary function lock mode during the machine lock mode, all auxiliary functions (spindle rotation, tool replacement, coolant on/off, etc.) are disabled. (See Section, "MACHINE LOCK AND AUXILIARY FUNCTION LOCK".)

1.5 EDITING A PROGRAM

After a created program is once registered in memory, it can be corrected or modified from the MDI panel. (See Chapter, "EDITING PROGRAMS".)
This operation can be executed using the program edit function.

1.6 DISPLAYING AND SETTING DATA

The operator can display or change a value stored in CNC internal memory by key operation on the MDI screen. (See Chapter, "SETTING AND DISPLAYING DATA".)

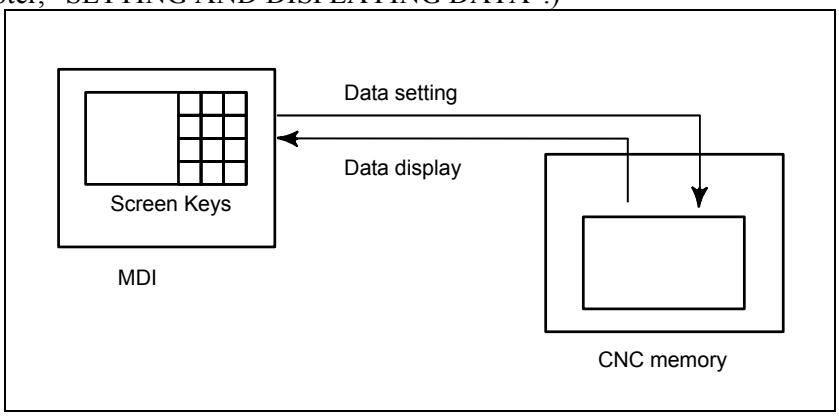


Fig. 1.6 (a) Displaying and setting data

Explanation

- Offset value

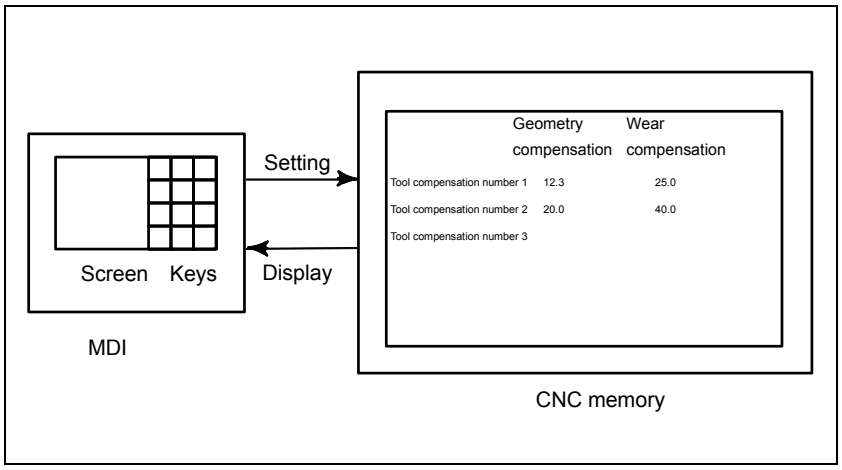


Fig. 1.6 (b) Displaying and Setting Offset Values

The tool has the tool dimension (length, diameter). When a workpiece is machined, the tool movement value depends on the tool dimensions.
By setting tool dimension data in CNC memory beforehand, CNC automatically generates tool routes that permit any tool to cut the workpiece specified by the program. Tool dimension data is called the offset value.

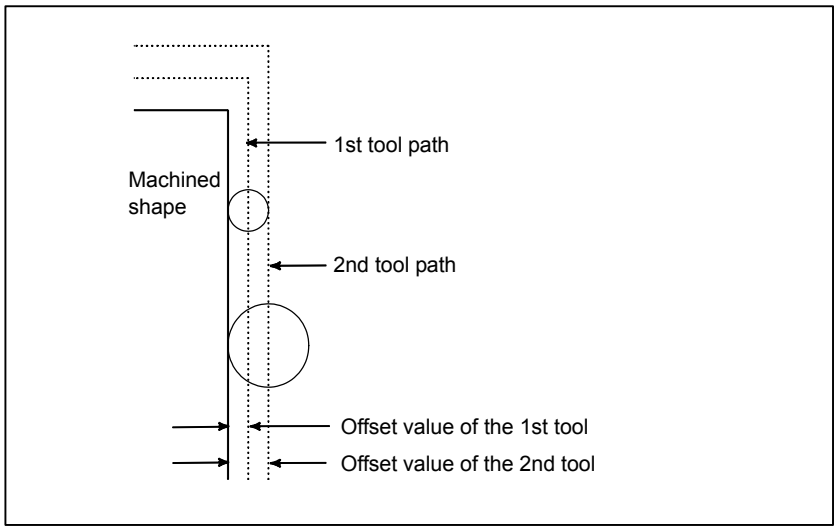


Fig. 1.6 (c) Offset value

- Displaying and setting operator's setting data

Apart from parameters, there is data that is set by the operator in operation. This data causes machine characteristics to change.
For example, the following data can be set:

- Inch/Metric switching
- Selection of I/O devices
- Mirror image cutting on/off

The above data is called setting data. (See Subsection, “Displaying and Entering Setting Data”.)

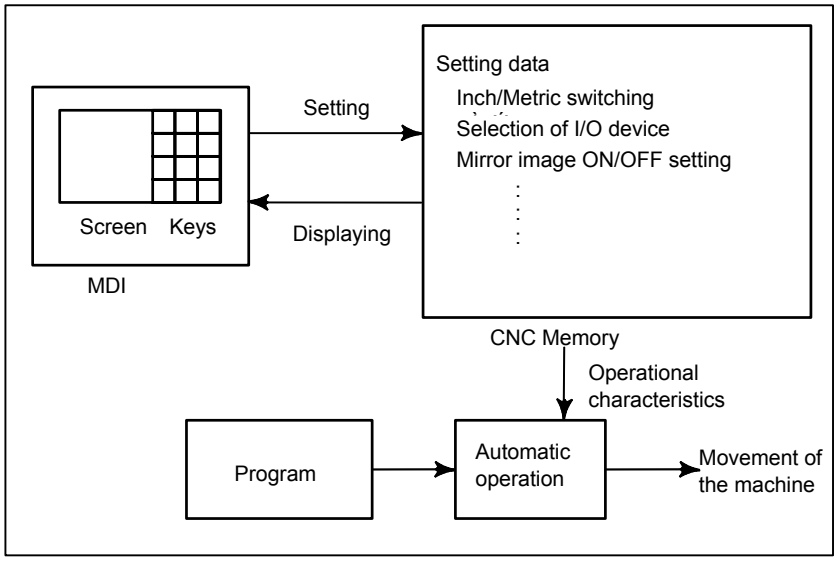


Fig. 1.6 (d) Displaying and setting operator's setting data

- Displaying and setting parameters

The CNC functions have versatility in order to take action in characteristics of various machines.
For example, CNC can specify the following:

- Rapid traverse rate of each axis
- Whether increment system is based on metric system or inch system.
- How to set command multiply/detect multiply (CMR/DMR)

Data to make the above specification is called parameters. (See Subsection, “Displaying and Setting Parameters”.)
Parameters differ depending on machine tool.

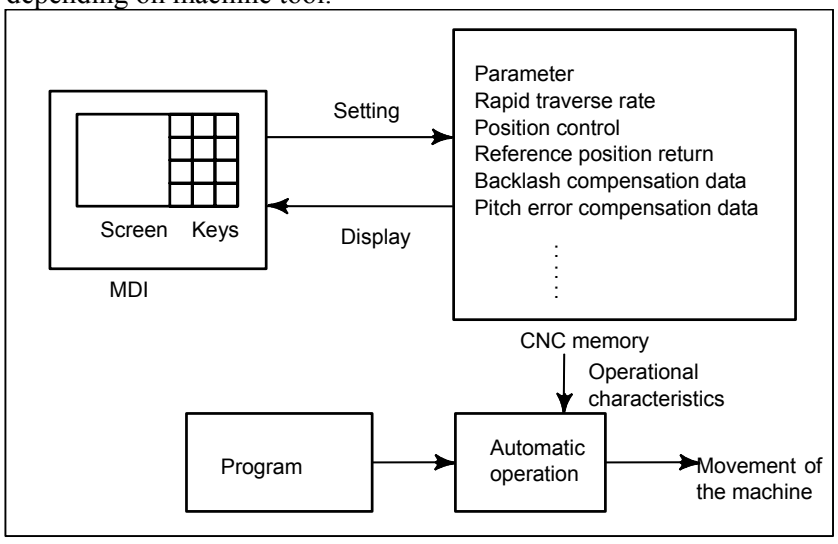


Fig. 1.6 (e) Displaying and setting parameters

- Data protection key

A key called the data protection key can be defined. It is used to prevent part programs, offset values, parameters, and setting data from being registered, modified, or deleted erroneously. (See Chapter, “SETTING AND DISPLAYING DATA”.)

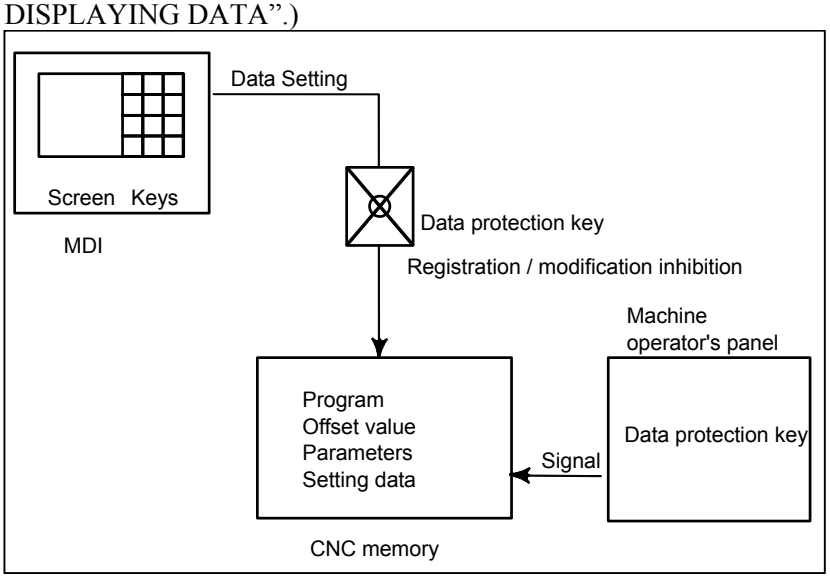


Fig. 1.6 (f) Data protection key

1.7 DISPLAY

1.7.1 Program Display

The contents of the currently active program are displayed.
(See Subsection, “Program Contents Display”.)

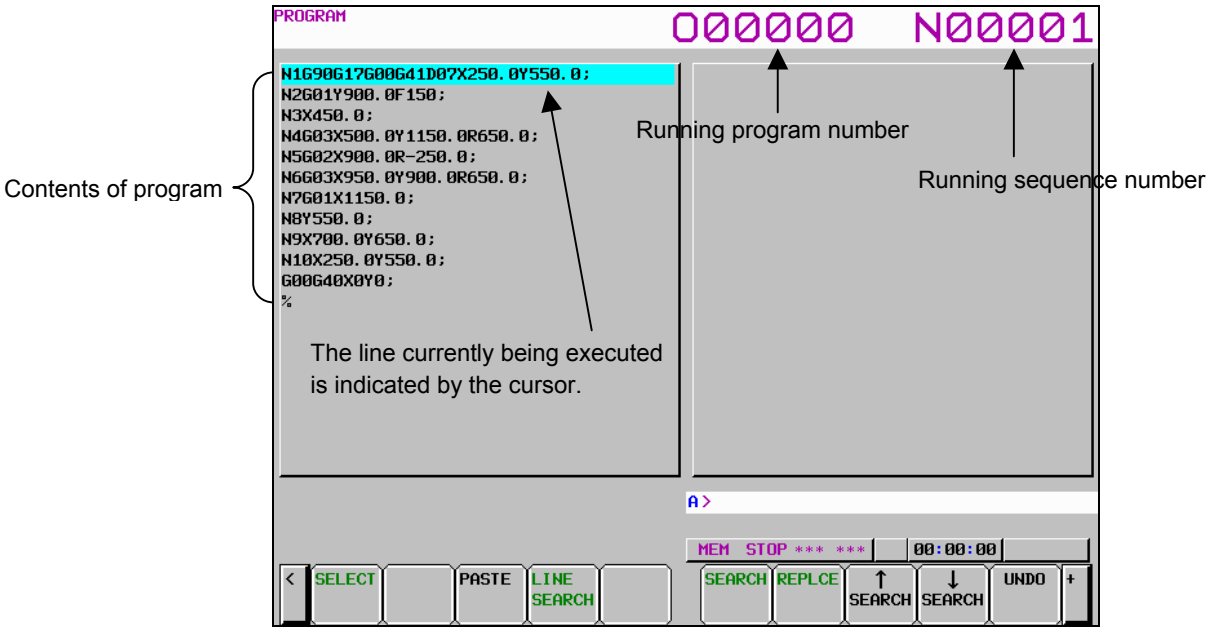


Fig. 1.7.1 (a)

A list of the programs held in the currently selected folder is displayed.

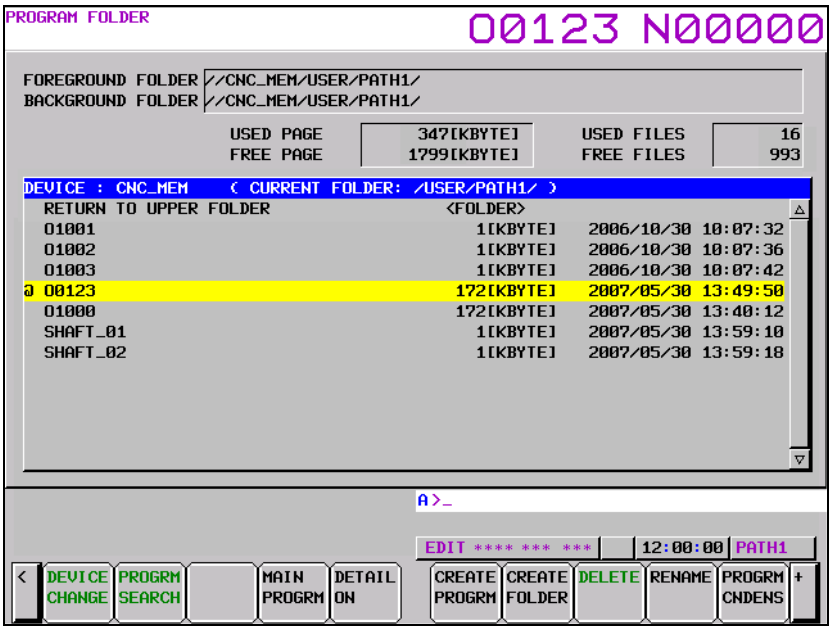


Fig. 1.7.1 (b)

1.7.2 Current Position Display

The current position of the tool is displayed with the coordinate values.
Moreover, the distance from the current position to a target point can be displayed as a remaining travel distance.
(See Subsections, “Position Display in the Workpiece Coordinate System”, “Position Display in the Relative Coordinate System”, and “Overall Position Display”.)

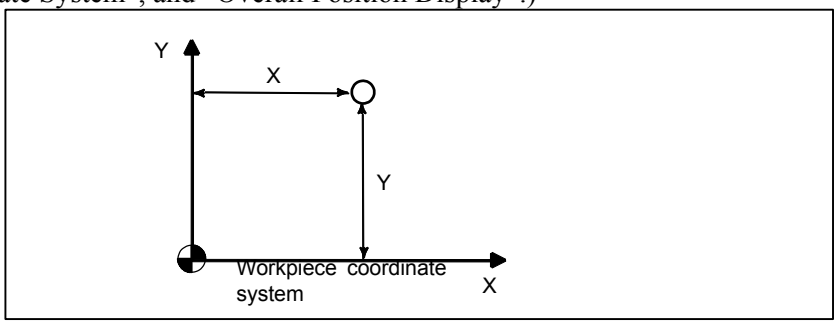


Fig. 1.7.2 (a)

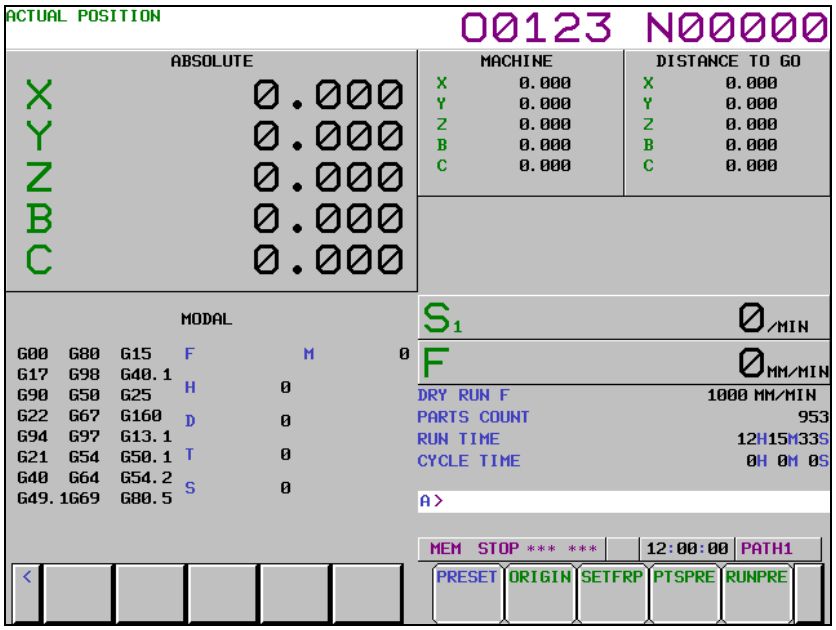


Fig. 1.7.2 (b)

1.7.3 Alarm Display

When a trouble occurs during operation, error code and alarm message are displayed on the screen. (See Section, “ALARM DISPLA”.)
See APPENDIX G for the list of error codes and their meanings.

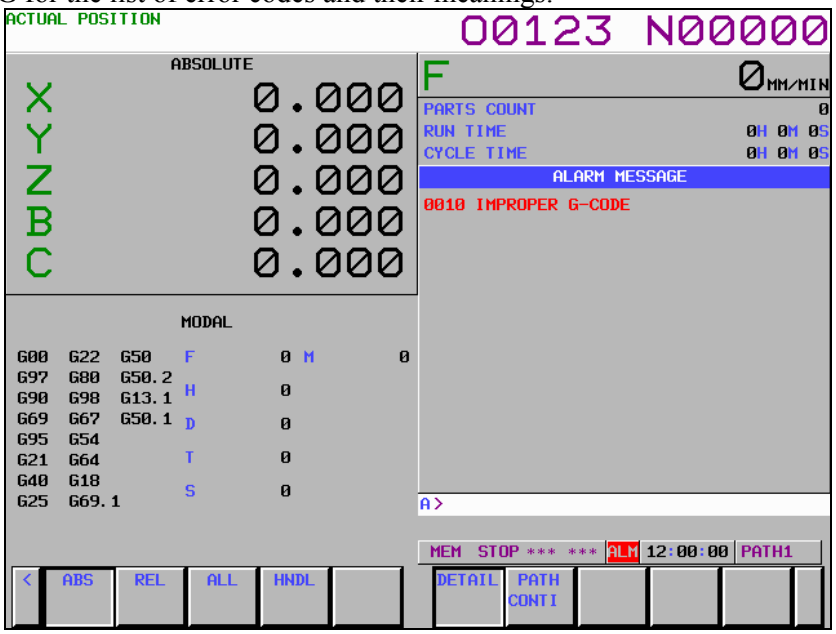


Fig. 1.7.3 (a)

1.7.4 Parts Count Display, Run Time Display

The position display screen displays a run time, cycle time, and parts count. (See Subsection, “Displaying and Setting Run Time, Parts Count, and Time”.)

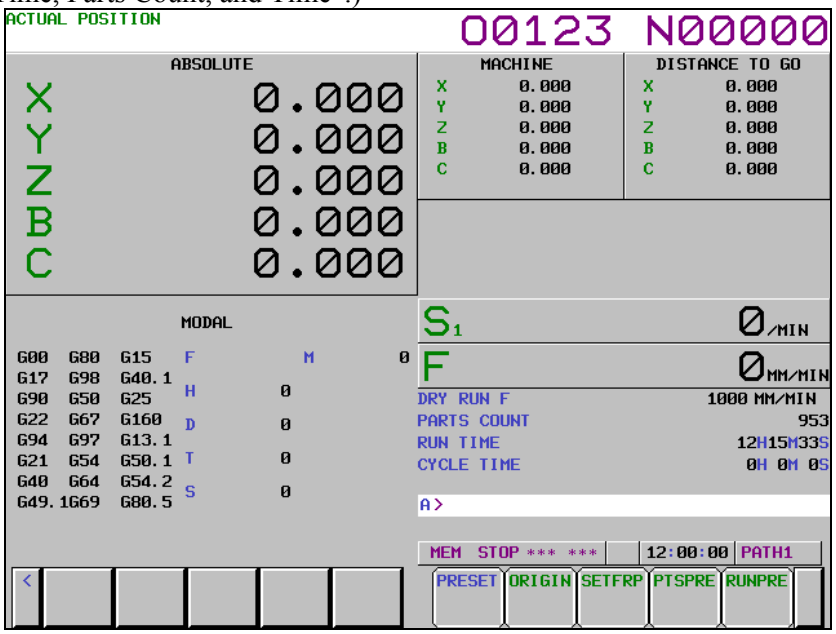


Fig. 1.7.4 (a)

2 OPERATIONAL DEVICES

As operational devices, setting and display devices attached to the CNC, and machine operator's panels are available.
For machine operator's panels, refer to the relevant manual of the machine tool builder.

Chapter 2, "OPERATIONAL DEVICES", consists of the following sections:

2.1 POWER ON/OFF.....937
2.1 SETTING AND DISPLAY UNITS.....939
2.3 EXPLANATION OF THE MDI UNIT.....944
2.4 FUNCTION KEYS AND SOFT KEYS.....947
2.5 EXTERNAL I/O DEVICES.....957

2.1 POWER ON/OFF

2.1.1 Turning on the Power

Procedure of turning on the power

Procedure

1 Check that the appearance of the CNC machine tool is normal.
(For example, check that front door and rear door are closed.)
2 Turn on the power according to the manual issued by the machine tool builder.
3 After the power is turned on, check that the position screen is displayed. An alarm screen is displayed if an alarm occurs upon power-on.

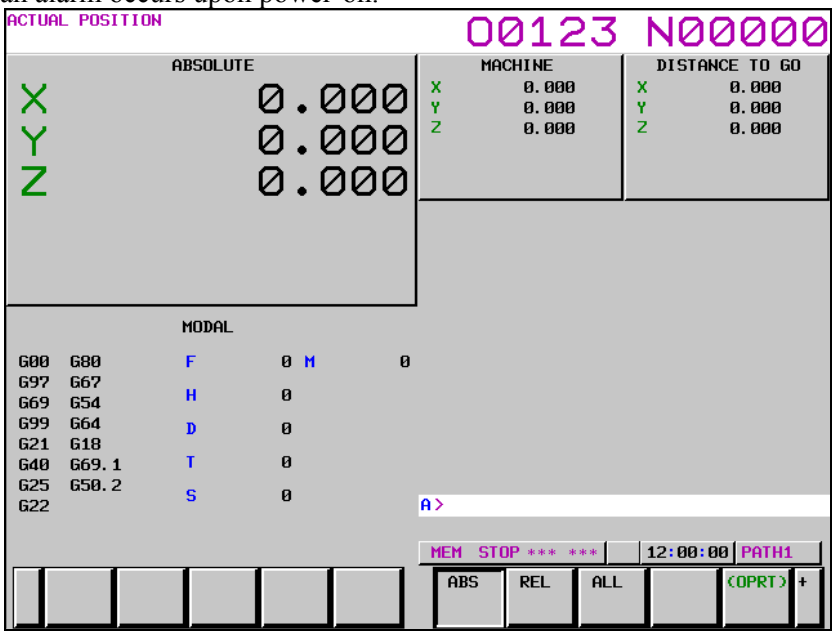


Fig. 2.1.1 (a) Position screen (for machining center system)

4 Check that the fan motor is rotating.

> ⚠ **WARNING**
> Until the positional or alarm screen is displayed at the power on, do not touch a key on the MDI unit. Some keys are used for the maintenance or special operation purpose. When they are pressed, unexpected operation may be caused.

2.1.2 Power Disconnection

Procedure of power disconnection

Procedure

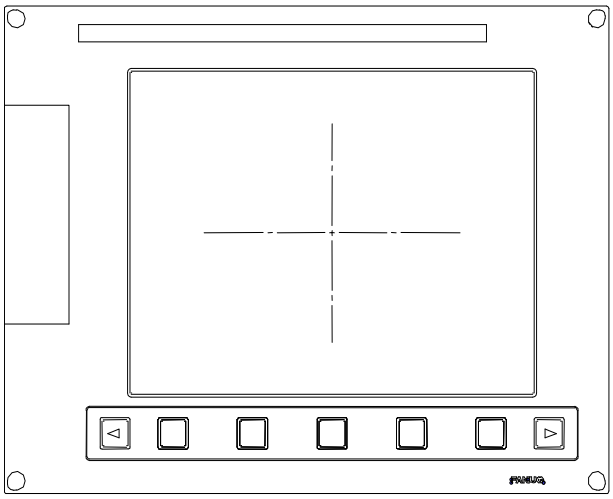
1 Check that the LED indicating the cycle start is off on the operator's panel.
2 Check that all movable parts of the CNC machine tool is stopping.
3 If an external input/output device such as the Handy File is connected to the CNC, turn off the external input/output device.
4 Continue to press the <POWER OFF> button for about 5 seconds.
5 Refer to the machine tool builder's manual for turning off the power to the machine.

2.2 SETTING AND DISPLAY UNITS

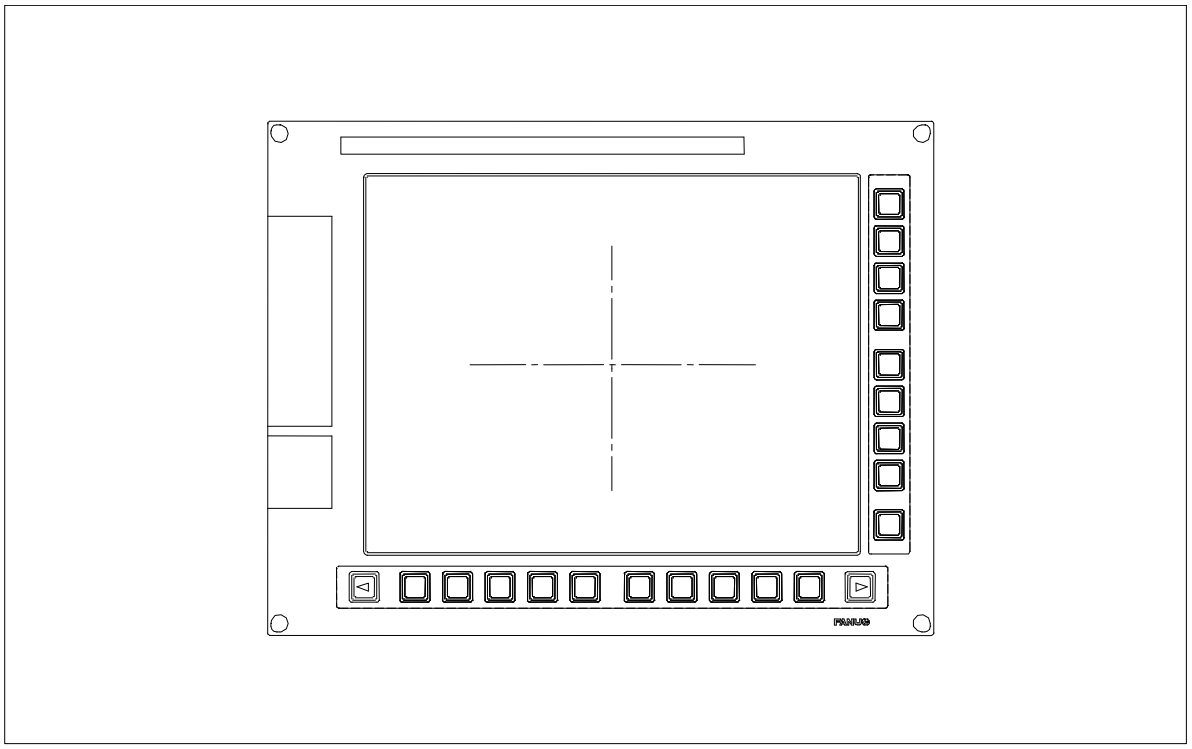
The setting and display units are shown in Subsections 2.1.1 to 2.1.8 of Part III.

8.4" LCD CNC Display Panel III-2.2.1
10.4" LCD CNC Display Panel (12.1"/15"/19" LCD CNC Display Panel) III-2.2.2
Standard MDI Unit (ONG Key) III-2.2.3
Standard MDI Unit (QWERTY Key) III-2.2.4
Small MDI Unit (ONG Key) III-2.2.5

2.2.1 8.4" LCD CNC Display Panel

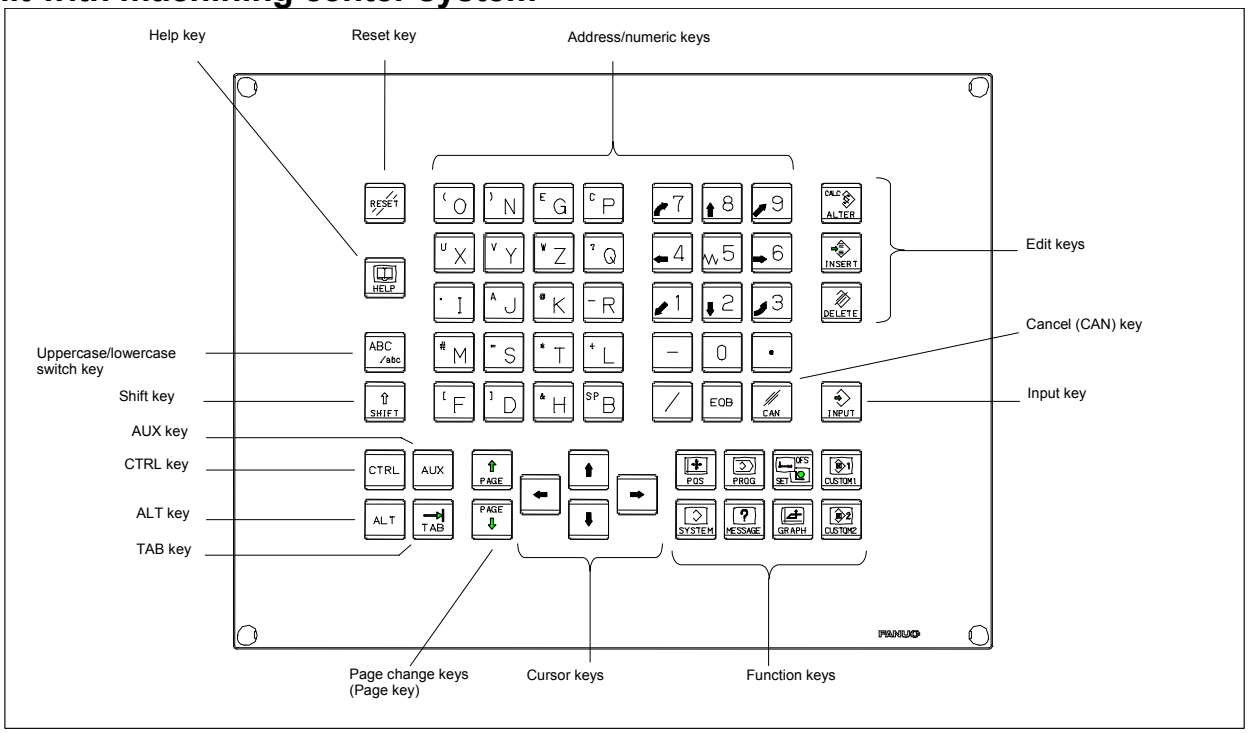


2.2.2 10.4" LCD CNC Display Panel (12.1"/15"/19" LCD CNC Display Panel)

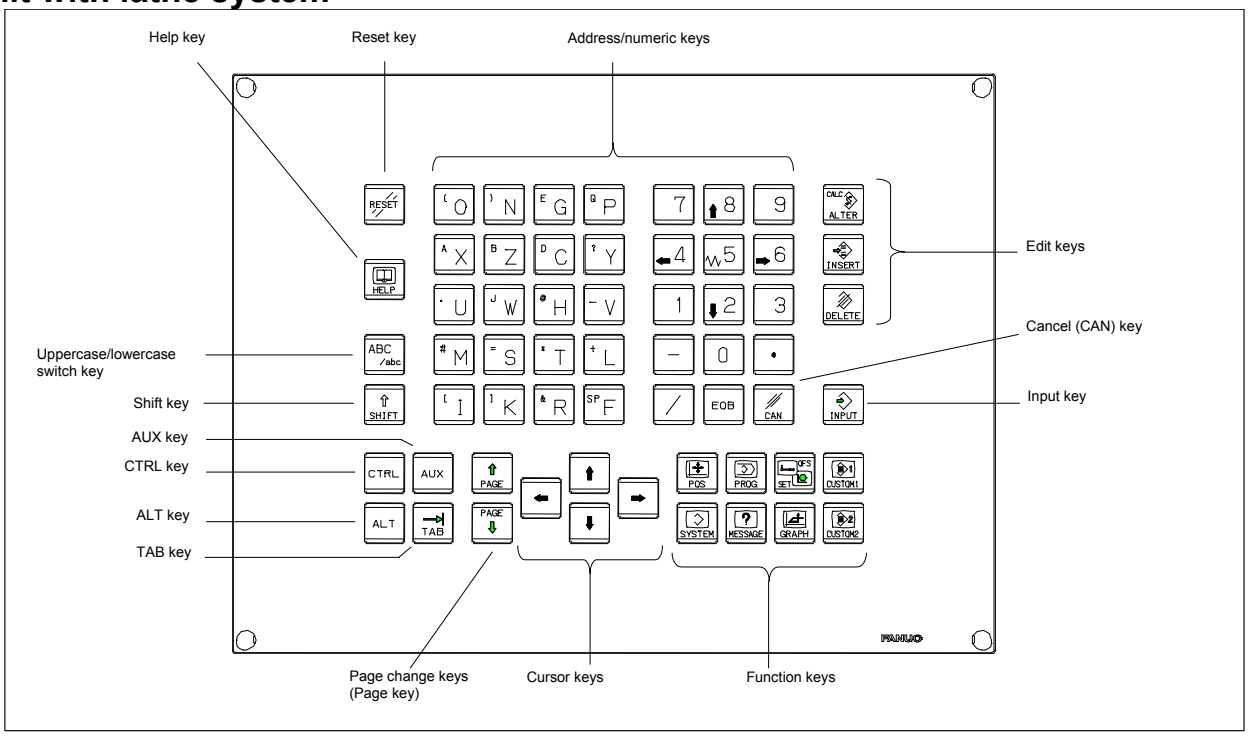


2.2.3 Standard MDI Unit (ONG Key)

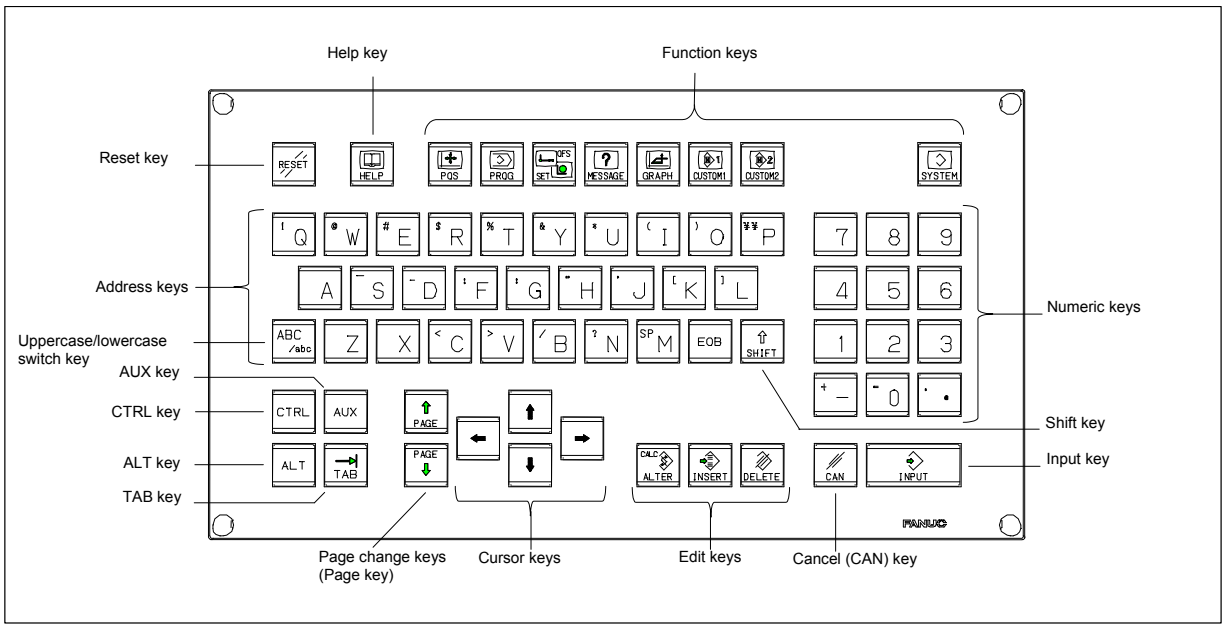
Unit with machining center system



Unit with lathe system

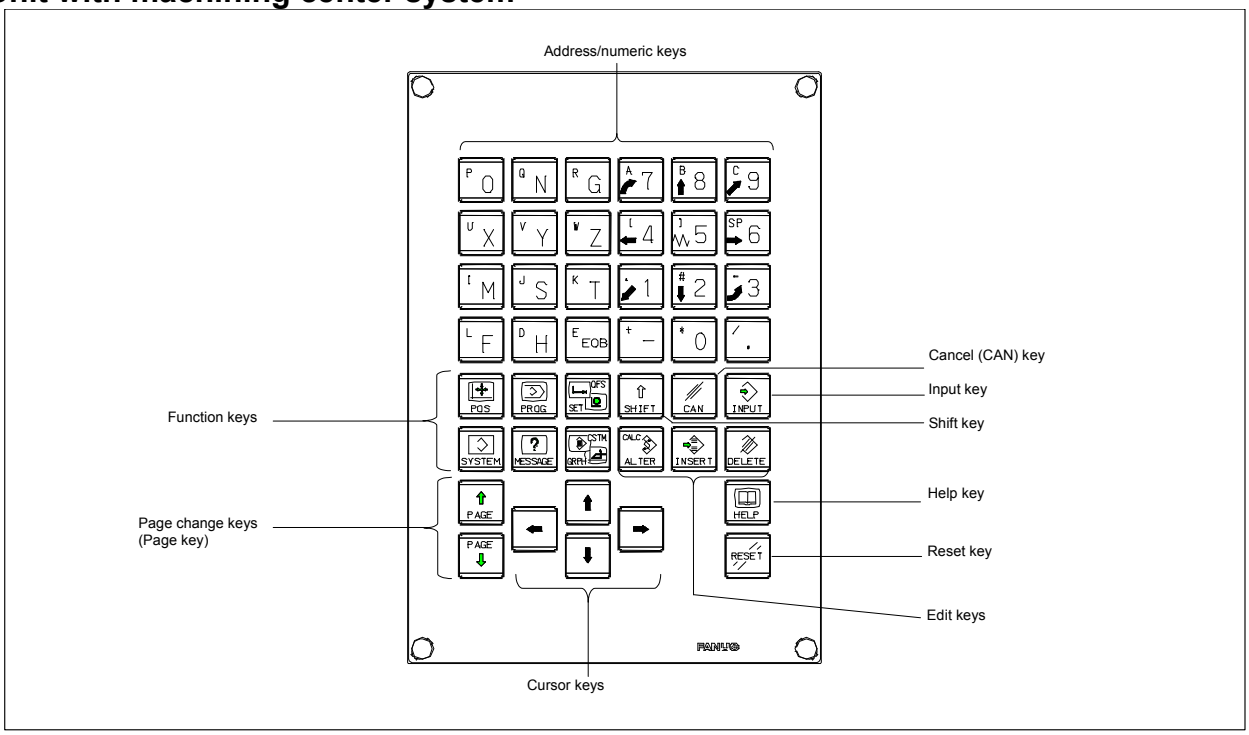


2.2.4 Standard MDI Unit (QWERTY Key)

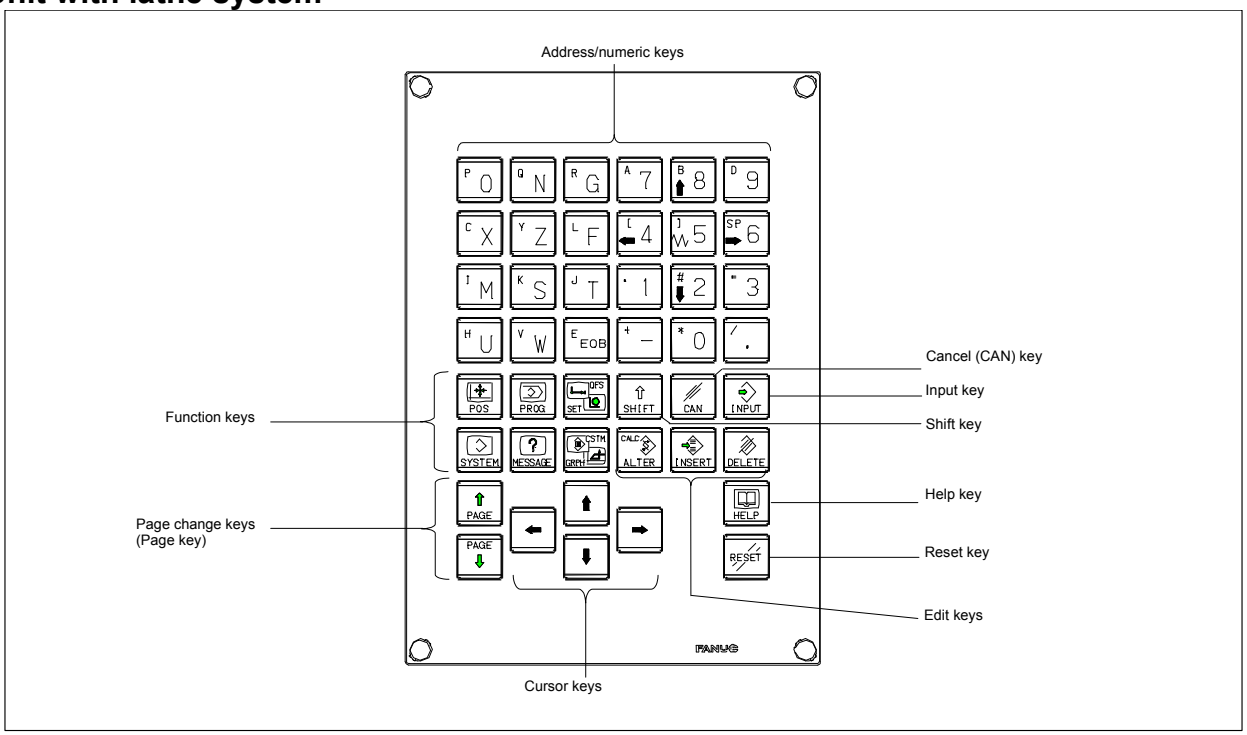


2.2.5 Small MDI Unit (ONG Key)

Unit with machining center system

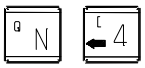
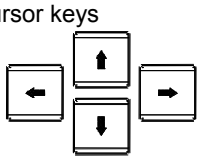


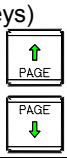
Unit with lathe system



2.3 EXPLANATION OF THE MDI UNIT

Table 2.3 (a) Explanation of the MDI unit

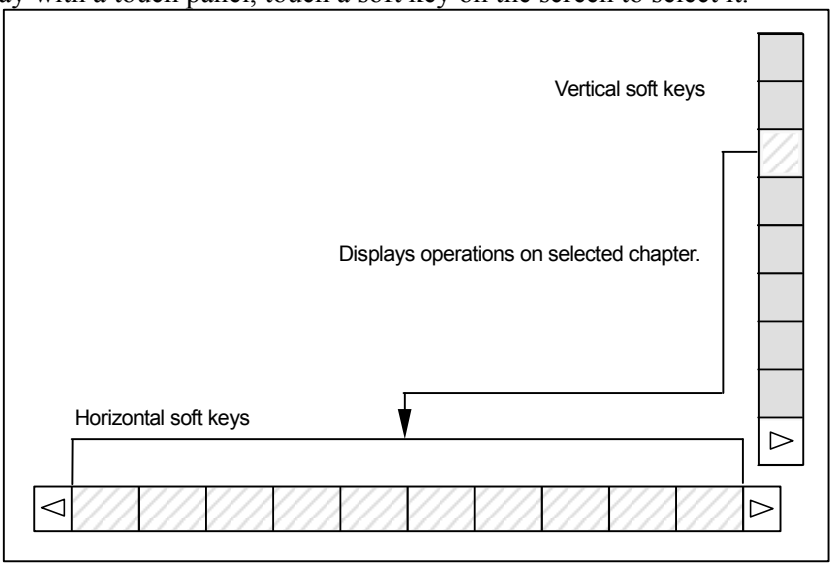
No.	Name	Explanation
1	RESET key 	Press this key to reset the CNC, to cancel an alarm, etc.
2	HELP key 	Press this key to use the help function when uncertain about the operation of an MDI key (help function).
3	Soft keys	The soft keys have various functions, according to the Applications. The soft key functions are displayed on the display unit.
4	Address and numeric keys 	Press these keys to input alphabetic, numeric, and other characters.
5	SHIFT key 	Some keys have two characters on their key top. Pressing the <SHIFT> key switches the characters. Special character ^ is displayed on the screen when a character indicated at the bottom right corner on the key top can be entered.
6	INPUT key 	When an address or a numerical key is pressed, the data is input to the buffer, and it is displayed on the screen. To copy the data in the key input buffer to the offset register, etc., press the  key. This key is equivalent to the [INPUT] key of the soft keys, and either can be pressed to produce the same result. This key is also used to move to a folder on the program folder screen.
7	CANCEL (CAN) key 	Press this key to delete the last character or symbol input to the key input buffer. Example) When the key input buffer displays >N001X100Z_ and the cancel key  is pressed, Z is canceled and >N001X100_ is displayed.
8	Edit keys 	Press these keys when editing the program.  : ALTER  : INSERT  : IDELETE
9	Function keys 	Press theses keys to switch display screens for each function. See Section, “Function Keys” for details of the function keys.
10	Cursor keys 	There are four different cursor move keys.  : This key is used to move the cursor to the right or in the forward direction. The cursor is moved in short units in the forward direction.  : This key is used to move the cursor to the left or in the reverse direction. The cursor is moved in short units in the reverse direction.  : This key is used to move the cursor in a downward or forward direction. The cursor is moved in large units in the forward direction.  : This key is used to move the cursor in an upward or reverse direction. The cursor is moved in large units in the reverse direction.

No.	Name	Explanation
11	Page change keys (Page keys) 	Two kinds of page change keys are described below.  : This key is used to changeover the page on the screen in the forward direction.  : This key is used to changeover the page on the screen in the reverse direction.
12	Uppercase/lowercase switch key 	Press this key to switch between uppercase and lowercase when entering alphabetic characters.
13	PC function key 	These keys are used with the personal computer functions.

Explanation

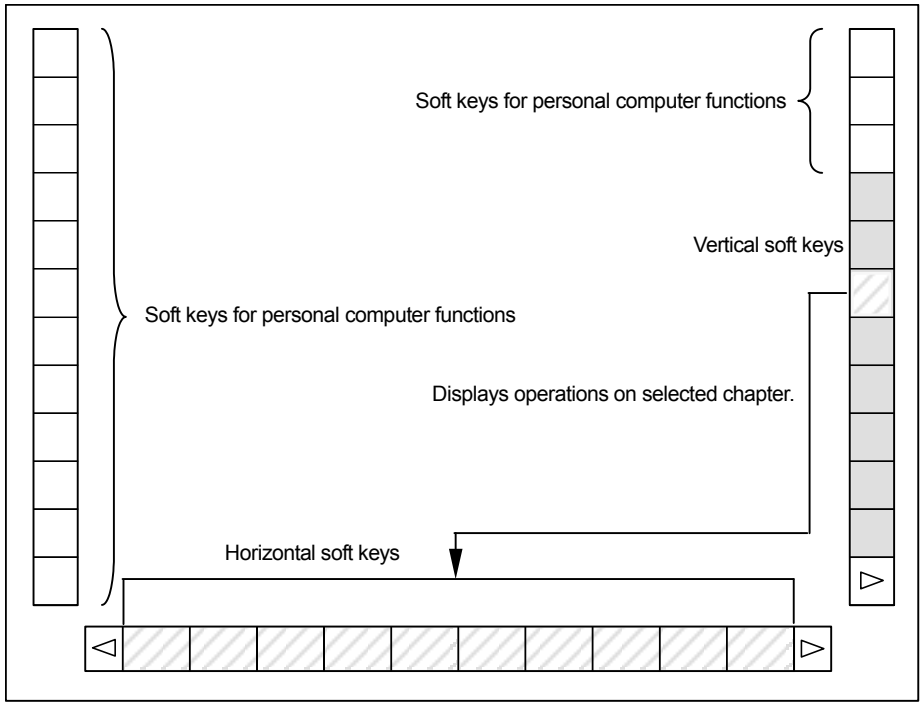
- 15” LCD/MDI soft key configuration

The 15” LCD panel has 12 soft keys horizontally and 9 soft keys vertically.
As shown below, the 8 vertical soft keys and the lowermost key are used as chapter selection soft keys. By pressing each of these keys, the screen (chapter) belonging to each function can be selected. The horizontal 12 soft keys are used to perform operations on the screen selected with a vertical soft key.
For an LCD display with a touch panel, touch a soft key on the screen to select it.



- 19” LCD/MDI soft key configuration

The 19” LCD panel has 12 soft keys horizontally and 12 soft keys vertically.
As shown below, the 9 lower vertical soft keys on the right side and the 12 horizontal soft keys are used for CNC screen operations and the other soft keys are available for personal computer functions.
Use the 9 lower vertical soft keys on the right side and 12 horizontal soft keys for CNC screen operations in the same way as for the 15” LCD panel.



In this manual, the descriptions below assume a 10.4" LCD display panel with 12 soft keys.

- MDI unit operation with multi-path control

In the multi-path control, be sure to select the tool post for which data is specified, using the path selection switch on the machine operator's panel. Then, perform MDI unit operation, such as displaying or specifying various data items, and editing a program.

2.4 FUNCTION KEYS AND SOFT KEYS

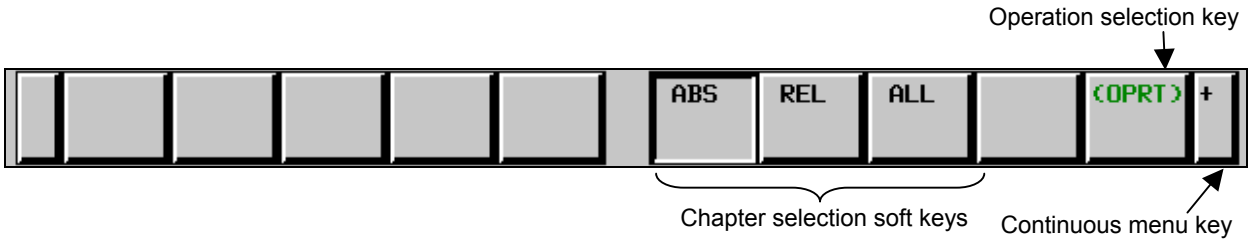
The function keys are used to select the type of screen (function) to be displayed. When a soft key (section select soft key) is pressed immediately after a function key, the screen (section) corresponding to the selected function can be selected.

2.4.1 General Screen Operations

- Procedure

1 By pressing a function key on the MDI unit, the chapter selection soft keys that belong to the function are displayed.

Example 1)

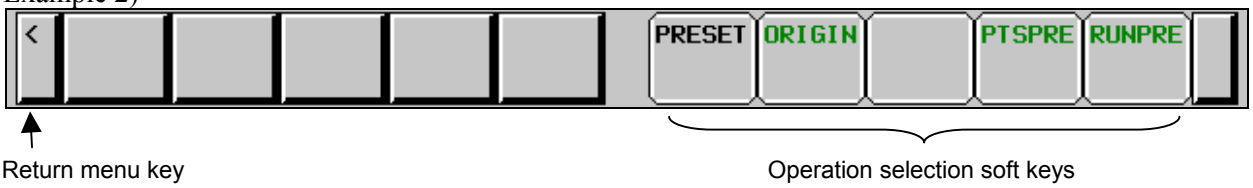


2 When one of the chapter selection soft keys is pressed, the screen of the chapter is displayed. If the soft key of a desired chapter is not displayed, press the continuous menu key.
In a chapter, a further choice may be made from multiple chapters.

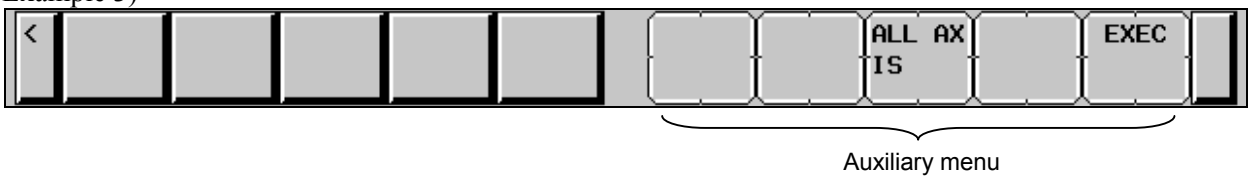
3 When the screen of a desired chapter is displayed, press the operation selection key to display operations to be performed.

4 Select a desired operation with the operation selection soft key.
Depending on the operation to be executed, an auxiliary menu of soft keys is displayed. Perform an operation according to the indications on the auxiliary menu.

Example 2)



Example 3)



5 To return to the display of chapter selection soft keys, press the return menu key.

A general screen display procedure is provided above.
The actual display procedure varies from one screen to another.
For details, see each description of operation.

- Button design change depending on soft key state

The soft keys assume one of the following states, depending on the selection target:

- Chapter selection soft keys
- Operation selection soft keys
- Auxiliary menu of operation selection soft keys

Depending on the state, the button images of the soft keys change.
From the button images, which state the soft keys are assuming can be known.

Example)

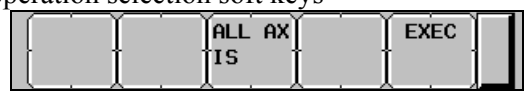
- Chapter selection soft keys



- Operation selection soft keys



- Auxiliary menu of operation selection soft keys



2.4.2 Function Keys

Function keys are provided to select the type of screen to be displayed. The following function keys are provided on the MDI unit:

Press this key to display the position screen.



Press this key to display the program screen.



Press this key to display the offset/setting screen.



Press this key to display the system screen.



Press this key to display the message screen.



Press this key to display the graphics screen.



Press this key to display the custom screen 1 (conversational macro screen or C Language Executor screen).



Press this key to display the custom screen 2 (conversational macro screen or C Language Executor screen).



2.4.3 Soft Keys

By pressing a soft key after a function key, the corresponding screen of the function can be displayed.

The chapter selection soft keys of each function are described below.
The horizontal four keys on the right-hand side are assigned to chapter selection soft keys. When multiple pages are used for chapter selection soft keys, [+] is displayed on the continuous menu key (rightmost soft key). Press the continuous menu key to switch between chapter selection soft keys.

NOTE
1 Press function keys to switch between screens that are used frequently.
2 Some soft keys are not displayed depending on the option configuration.

If position indications are provided on the left half of the screen when a key other than the function key 

is pressed, the left half of the soft keys is displayed as follows at all times:



Position display screen

The chapter selection soft keys that belong to the function key 

and the function of each screen are described below.

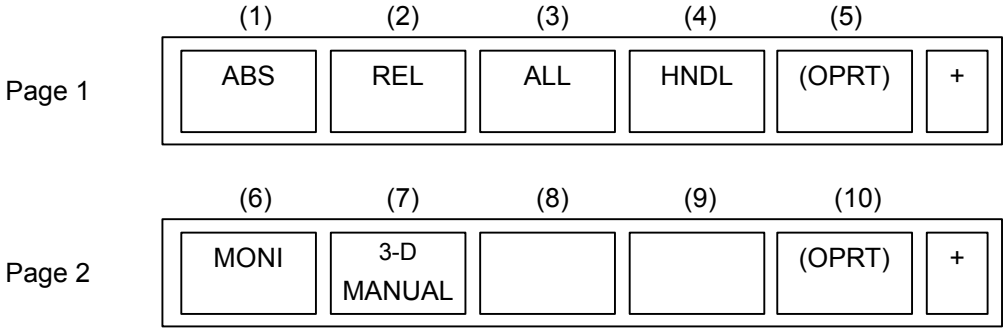


Table 2.4.3 (a) Position display screen

No.	Chapter menu	Description
(1)	ABS	Selects the absolute coordinate display screen.
(2)	REL	Selects the relative coordinate display screen.
(3)	ALL	Selects the overall coordinate display screen.
(4)	HNDL	Selects the operation screen for manual handle operation.

No.	Chapter menu	Description
(6)	MONI	Selects the screen for displaying the servo axis load meter, serial spindle load meter, and speedometer.
(7)	3-D MANUAL	Displays a handle pulse interrupt amount in 3-dimensional manual feed.

Program screen

The chapter selection soft keys that belong to the function key  and the function of each screen are described below.

Table 2.4.3 (b) Program

No.	Chapter menu	Description
(1)	PROGRAM	Selects the screen for displaying a list of part programs currently registered.
(2)	FOLDER	Selects the screen for displaying a list of part programs currently registered.
(3)	NEXT	Selects the screen for displaying the command values of the block currently executed and the next block to be executed among the command values.
(4)	CHECK	Selects the screen for displaying programs, position data, modal information, and so forth simultaneously.
(6)	TIME	Selects the screen for displaying executed program operation time.
(7)	JOG	Selects the screen for executing, in the JOG mode, data specified in the program format from the MDI.
(8)	RSTR	Selects the operation screen for restarting an interrupted program operation.
(9)	ROBOT SELECT	Selects the screen for registering robot programs and part programs for the Robot connection function.

Offset/setting screen

The chapter selection soft keys that belong to the function key  and the function of each screen are described below.

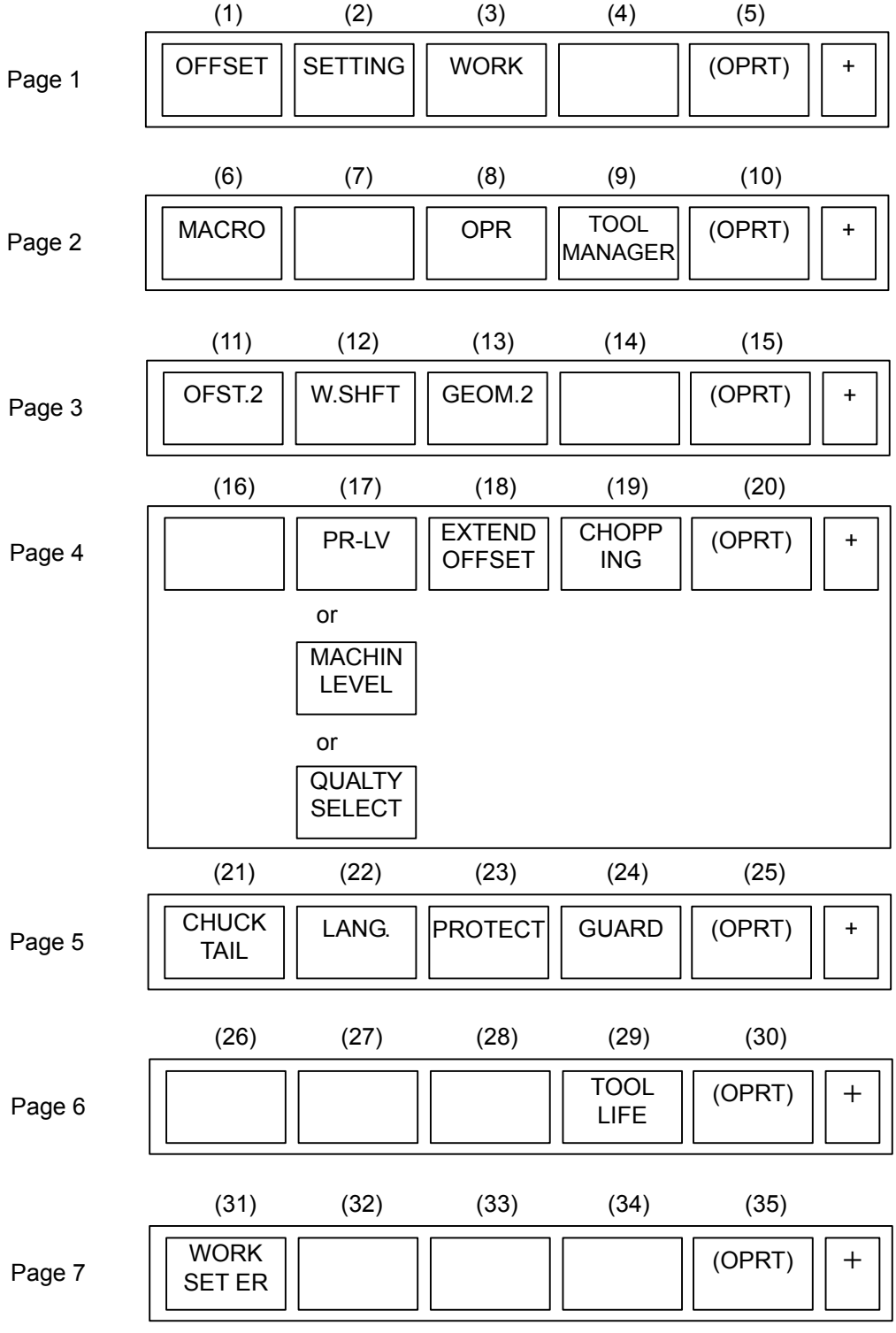


Table 2.4.3 (c) Offset

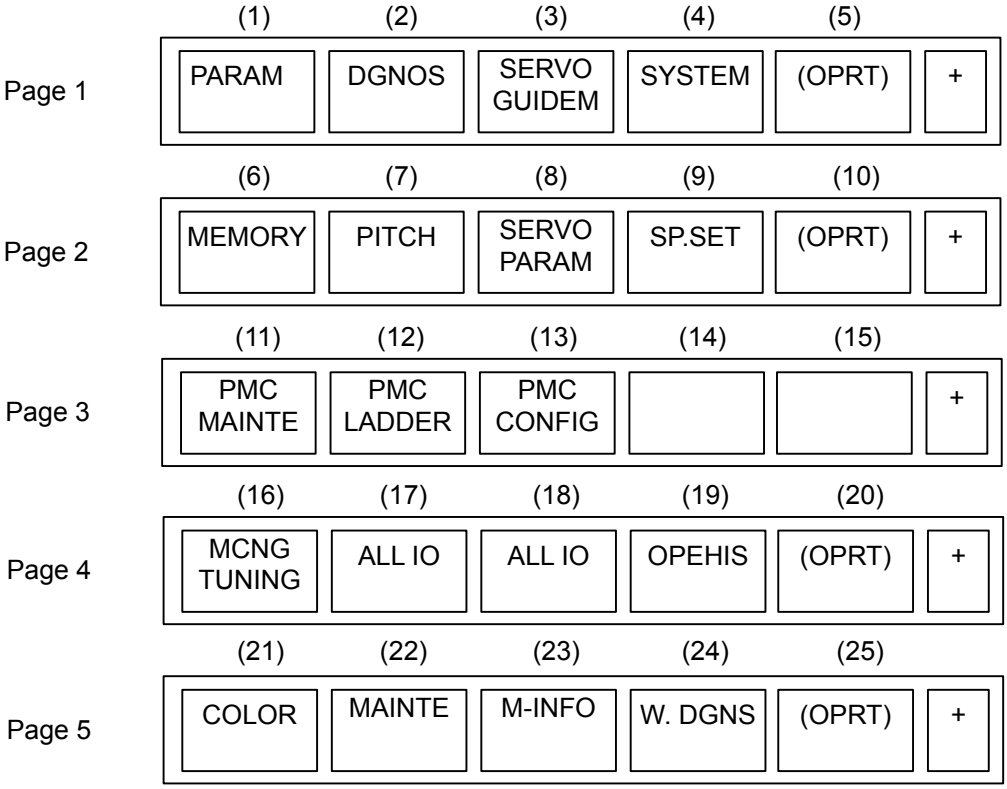
No.	Chapter menu	Description
(1)	OFFSET	Selects the screen for setting tool offset values.
(2)	SETTING	Selects the screen for setting the setting parameters.
(3)	WORK	Selects the screen for setting a workpiece coordinate system offset.

No.	Chapter menu	Description
(6)	MACRO	Selects the screen for setting macro variables.
(8)	OPR	Selects the screen for operating some operation switches on the machine operator's panel as soft switches.
(9)	TOOL MANAGER	Selects the screen for setting data related to tool management.
(11)	OFST.2	Selects the screen for setting a Y-axis offset.
(12)	W.SHFT	Selects the screen for setting a workpiece coordinate system shift value.
(13)	GEOM.2	Selects the screen for setting a second geometry offset.
(17)	PR-LV	Selects the screen for setting a precision level.
(18)	EXTEND OFFSET	Selects the screen for setting the offsets of the fourth and fifth axes.
(21)	CHUCK TAIL	Selects the chuck tail stock barrier screen.
(22)	LANG.	Selects the screen for setting a display language.
(23)	PROTECT	Selects the screen for setting data protection.
(24)	GUARD	Selects the screen for setting wrong operation prevention.
(29)	TOOL LIFE	Selects the screen for operations and setting related to tool life management.
(31)	WRK ERR COMP	Selects the screen for setting errors related to workpiece mounting position.

System screen

The chapter selection soft keys that belong to the function key 

and the function of each screen are described below.



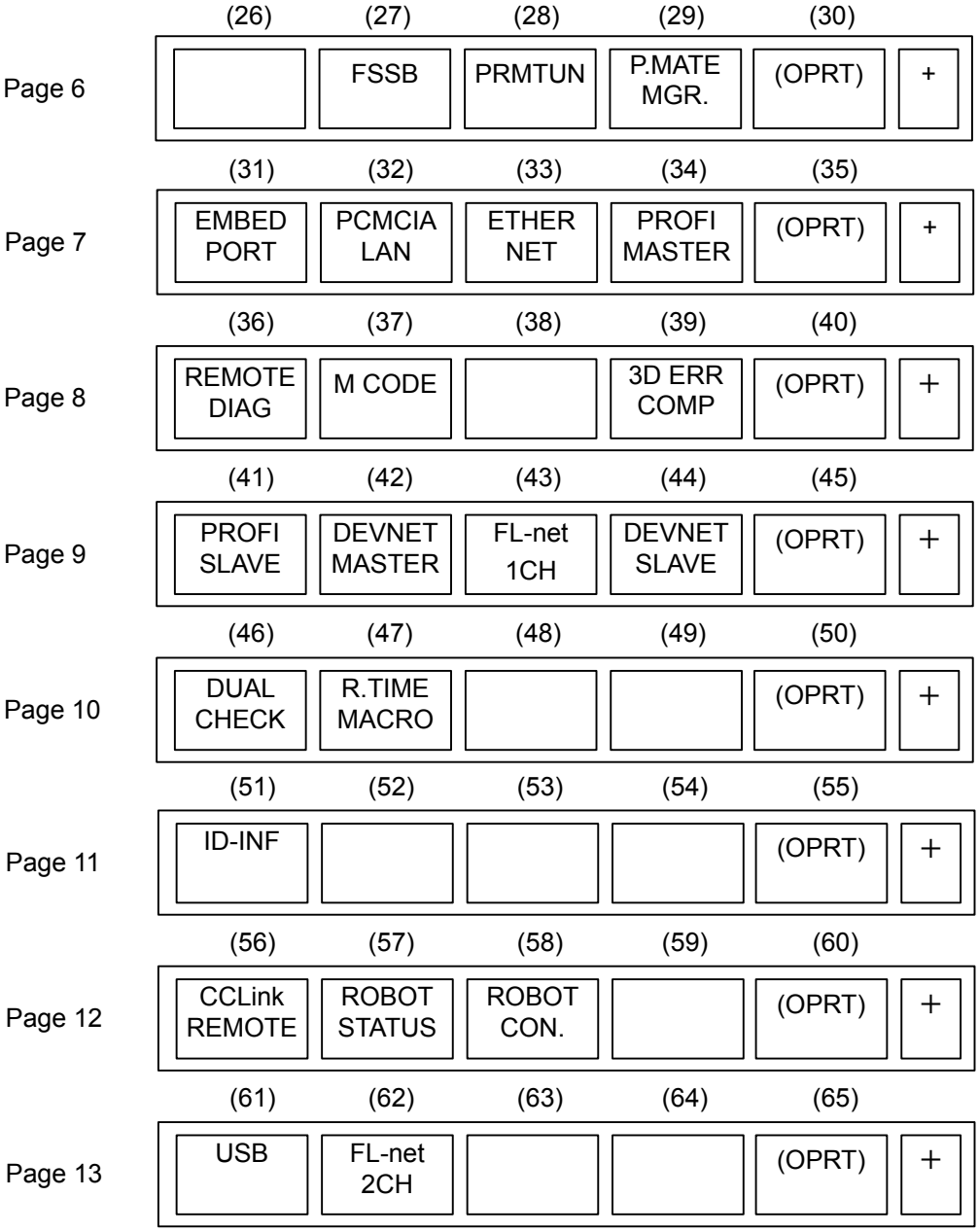


Table 2.4.3 (d) System

No.	Chapter menu	Description
(1)	PARAM	Selects the screen for setting parameters.
(2)	DGNOS	Selects the screen for displaying CNC state.
(3)	SERVO GUIDEM	Selects the screen for displaying the servo guide mate.
(4)	SYSTEM	Selects the screen for displaying the current system status.
(6)	MEMORY	Selects the screen for displaying the contents of memory.
(7)	PITCH	Selects the screen for setting pith error compensation.
(8)	SERVO PARAM	Selects the screen for setting the servo-related parameters.
(9)	SP.SET	Selects the screen for spindle-related setting.
(11)	PMC MAINTE	Selects the screen related to PMC maintenance such as PMC signal state monitoring and tracing, and PMC parameter display/editing.
(12)	PMC LADDER	Selects the screen related to ladder display/editing.
(13)	PMC CONFIG	Displays the screen for displaying/editing data other than ladders that makes up a sequence program and for setting the PMC function.

No.	Chapter menu	Description
(16)	MCNG TUNING	Displays the screen for setting the parameter set for emphasis on speed (LV1) or emphasis on precision (LV10).
(17)	ALL IO (RS232C interface)	Selects the screen for data input to and output from the memory card.
(18)	ALL IO (Memory card)	Selects the screen for data input to and output from the memory card.
(19)	OPEHIS	Selects the screen for displaying the history of operations performed by the operator and issued alarms.
(21)	COLOR	Selects the screen for setting colors to be used on the screen.
(22)	MAINTE	Selects the screen for setting maintenance items to be managed periodically.
(23)	M-INFO	Selects the screen for displaying information about maintenance performed.
(24)	W.DGNS	Selects the screen for displaying data such as servo positional deviation values, torque values, machine signals, and so forth as graphs.
(27)	FSSB	Selects the screen for making settings related to the high-speed serial servo bus (FSSB: Fanuc Serial Servo Bus).
(28)	PRMTUN	Selects the screen for setting parameters necessary for start-up and tuning.
(31)	EMBED PORT	Selects the screen for making settings related to the embedded Ethernet (embedded port).
(32)	PCMCIA LAN	Selects the screen for making settings related to the embedded Ethernet (PCMCIA Ethernet card).
(33)	ETHERNET	Selects the screen for making settings related to the fast Ethernet/fast data server.
(34)	PROFI MASTER	Selects the screen for making settings related to the PROFIBUS master function.
(36)	REMOTE DIAG	Selects the screen for making settings related to the Machine Remote Diagnosis.
(37)	M CODE	Selects the screen for setting an M code group.
(41)	PROFI SLAVE	Selects the screen for making settings related to the PROFIBUS slave function.
(42)	DEVNET MASTER	Selects the screen for making settings related to the DeviceNet master function.
(43)	FL-net 1CH	Selects the screen for making settings related to the FL-net (port 1) function.
(44)	DEVNET SLAVE	Selects the screen for making settings related to the DeviceNet slave function.
(51)	ID-INF	Selects the screen for displaying CNC ID information.
(56)	CCLink REMOTE	Selects the screen for making settings related to the CC-Link remote device function.
(57)	ROBOT STATUS	Selects the robot status screen for the Robot connection function.
(58)	ROBOT CON.	Selects the screen for making settings related to the Robot connection function.
(61)	USB	Selects the USB maintenance screen.
(62)	FL-net 2CH	Selects the screen for making settings related to the FL-net (port 2) function.

Message screen

The chapter selection soft keys that belong to the function key  and the function of each screen are described below.

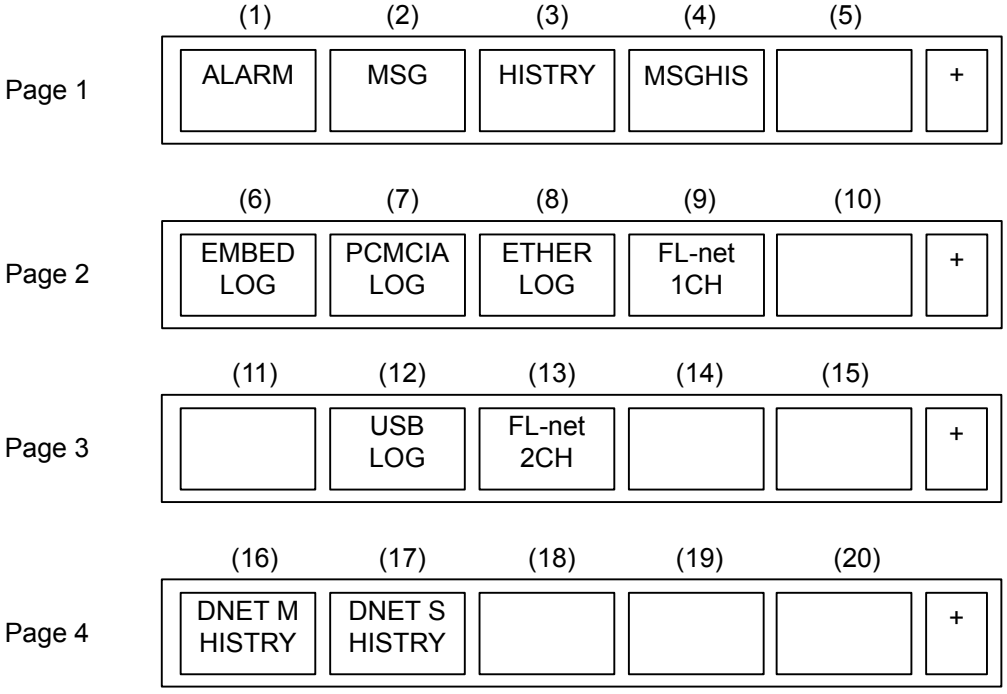


Table 2.4.3 (e) Message

No.	Chapter menu	Description
(1)	ALARM	Selects the alarm message screen.
(2)	MSG	Selects the operator message screen.
(3)	HISTRY	Selects the screen for displaying the details of alarms issued so far.
(4)	MSGHIS	Selects the external operator message screen.
(6)	EMBED LOG	Selects the screen for displaying error messages related to the embedded Ethernet (embedded port).
(7)	PCMCIA LOG	Selects the screen for displaying error messages related to the embedded Ethernet (PCMCIA Ethernet card).
(8)	ETHER LOG	Selects the screen for displaying error messages related to the fast Ethernet/fast data server.
(9)	FL-net 1CH	Selects the screen for displaying error messages related to the FL-net (port 1) function.
(12)	USB LOG	Selects the screen for displaying error messages related to the USB function.
(13)	FL-net 2CH	Selects the screen for displaying error messages related to the FL-net (port 2) function.
(16)	DNET M HISTRY	Selects the screen for displaying communication histories related to the DeviceNet master function.
(17)	DNET S HISTRY	Selects the screen for displaying communication histories related to the DeviceNet slave function.

Graphic screen

The chapter selection soft keys that belong to the function key  and the function of each screen are described below.

When the graphic display function is enabled:

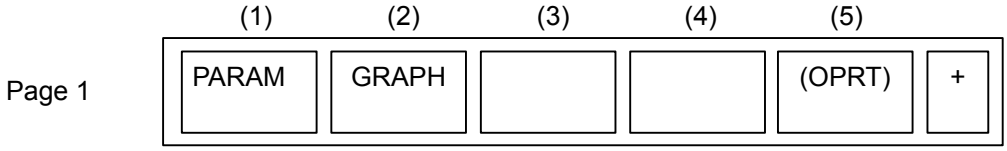


Table 2.4.3 (f) Graphic

No.	Chapter menu	Description
(1)	PARAM	Selects the screen for setting graphic parameters.
(2)	GRAPH	Selects the screen for graphically displaying the tool path.

When the dynamic graphic display function is enabled:

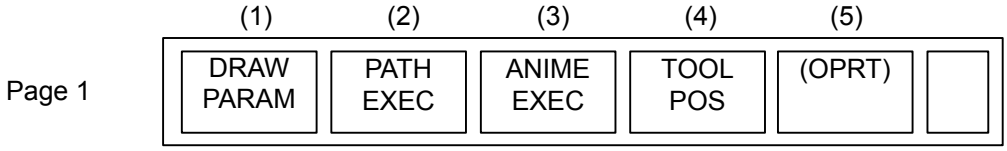


Table 2.4.3 (g) Graphic

No.	Chapter menu	Description
(1)	DRAW PARAM	Selects the screen for setting drawing parameters.
(2)	PATH EXEC	Selects the screen for drawing tool paths.
(3)	ANIME EXEC	Selects the screen for drawing animation.
(4)	TOOL POS	Selects the screen for displaying tool positions.

2.5 EXTERNAL I/O DEVICES

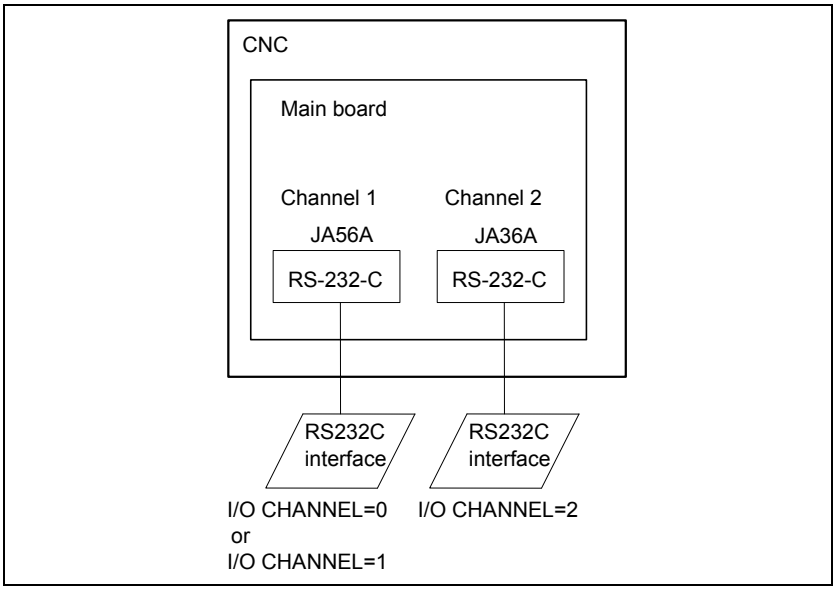
External I/O devices such as a memory card are available.
By using an external I/O device such as a memory card, the following data can be input or output:

1. Programs
2. Offset data
3. Parameters
4. Custom macro common variables

For how to input or output data and input data from or output it to a memory card, see Chapter, "DATA INPUT/OUTPUT".

- Parameter setting

Before an external input/output device can be used, parameters must be set as follows.



This CNC has a total of two channels of RS232C interfaces. It also has memory card interface and USB interface. The input/output device to be used is specified by setting the channel (interface) connected to that device in setting parameter I/O CHANNEL.
The specified data, such as a baud rate and the number of stop bits, of an input/output device connected to a specific channel must be set in parameters for that channel in advance. (These settings are not required for the memory card interface and USB interface.)
For channel 1, two combinations of parameters to specify the input/output device data are provided.
The following shows the interrelation between the RS232C interface parameters for the channels.

0020 I/O CHANNEL
or foreground input

Set channels to be used for data input/output.

I/O CHANNEL (0 to 5)
=0 : Channel 1
=1 : Channel 1
=2 : Channel 2
=3 : Channel 3
:
:
:

Input/output to and from the memory card interface, etc. is also possible.

When IO4 is set

0021 Foreground output

0022 Background input

0023 Background input

The channel setting is the same as No.0020.

Input/output channel number (parameter No.0020)

↓

I/O CHANNEL=0 (Channel 1)
- 0101 Stop bit and other data
- 0102 Number specified for the input/output device
- 0103 Baud rate

I/O CHANNEL=1 (Channel 1)
- 0111 Stop bit and other data
- 0112 Number specified for the input/output device
- 0113 Baud rate

I/O CHANNEL=2 (Channel 2)
- 0121 Stop bit and other data
- 0122 Number specified for the input/output device
- 0123 Baud rate

:
:
:

I/O CHANNEL=5

3 MANUAL OPERATION

MANUAL OPERATION are twelve kinds as follows :

3.1 MANUAL REFERENCE POSITION RETURN 959
3.2 JOG FEED (JOG) 960
3.3 INCREMENTAL FEED 962
3.4 MANUAL HANDLE FEED 963
3.5 MANUAL ABSOLUTE ON AND OFF 966
3.6 MANUAL LINEAR/CIRCULAR INTERPOLATION 970
3.7 RIGID TAPPING BY MANUAL HANDLE 974
3.8 MANUAL NUMERICAL COMMAND 976
3.9 3-DIMENSIONAL MANUAL FEED 984
3.10 DISTANCE CODED LINEAR SCALE INTERFACE 997
3.11 LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL) 1002

3.1 MANUAL REFERENCE POSITION RETURN

The tool is returned to the reference position as follows :
The tool is moved in the direction specified in bit 5 (ZMI) of parameter No.1006 for each axis with the reference position return switch on the machine operator's panel. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed. The rapid traverse rate and FL speed are specified in parameters Nos. 1424,1421, and 1425.
Four step rapid traverse override is effective during rapid traverse.
When the tool has returned to the reference position, the reference position return completion LED goes on. The tool generally moves along only a single axis, but can move along three axes simultaneously when specified so in bit 0 (JAX) of parameter No.1002.

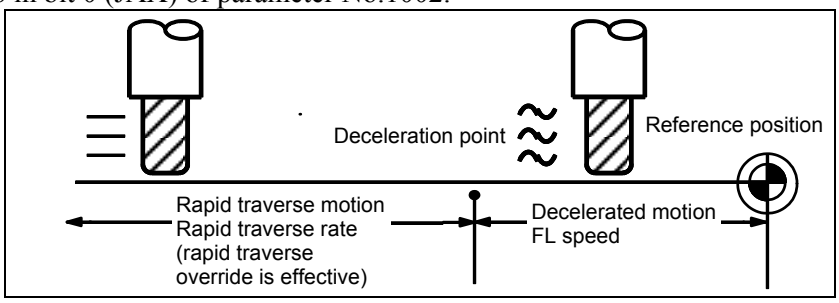


Fig. 3.1 (a) Manual reference position return

Procedure for manual reference position return

Procedure

1 Press the reference position return switch, one of the mode selection switches.
2 To decrease the feedrate, press a rapid traverse override switch.
3 Press the feed axis and direction selection switch corresponding to the axis and direction for reference position return. Continue pressing the switch until the tool returns to the reference position. The tool can be moved along three axes simultaneously when specified so in an appropriate parameter setting. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed set in a parameter. When the tool has returned to the reference position, the reference position return completion LED goes on.
4 Perform the same operations for other axes, if necessary.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

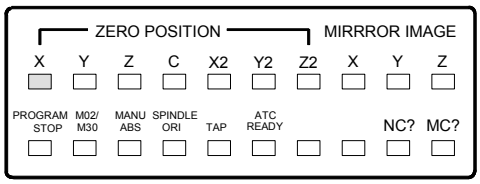


Fig. 3.1 (b)

Explanation

- Automatically setting the coordinate system

Bit 0 (ZPR) of parameter No.1201 is used for automatically setting the coordinate system. When ZPR is set, the coordinate system is automatically determined when manual reference position return is performed.

When α, β and γ are set in parameter 1250, the workpiece coordinate system is determined so that reference point on the tool holder or the position of the tip of the reference tool is X= α, Y = β, Z = γ when reference position return is performed. This has the same effect as specifying the following command for reference position return:

G92XαYβZγ;

However, when options of the workpiece coordinate system is selected, it is not able to use.

Limitation

- Moving the tool again

Once the reference position return completion LED lights at the completion of reference position return, the tool does not move unless the reference position return switch is turned off.

- Reference position return completion LED

The reference position return completion LED is extinguished by either of the following operations:

- Moving from the reference position.
- Entering an emergency stop state.

- The distance to return to reference position

For the distance (Not in the deceleration condition) to return the tool to the reference position, refer to the manual issued by the machine tool builder.

3.2 JOG FEED (JOG)

In the jog mode, pressing a feed axis and direction selection switch on the machine operator's panel continuously moves the tool along the selected axis in the selected direction.

The jog feedrate is specified in a parameter No.1423.

The jog feedrate can be adjusted with the jog feedrate override dial.

Pressing the rapid traverse switch moves the tool at the rapid traverse feedrate No. 1424 regardless of the position of the jog feedrate override dial. This function is called the manual rapid traverse.

Manual operation is allowed for one axis at a time. 3 axes can be selected at a time by bit 0 (JAX) of parameter No.1002.

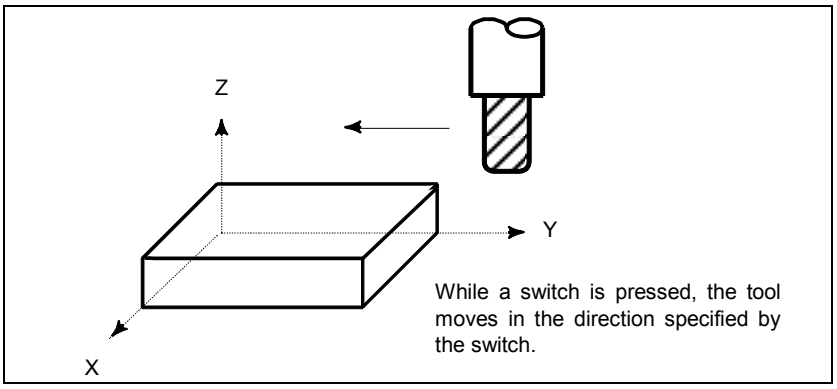


Fig. 3.2 (a) Jog Feed (JOG)

Procedure for JOG feed

Procedure

1 Press the jog switch, one of the mode selection switches.
2 Press the feed axis and direction selection switch corresponding to the axis and the direction where the tool is to be moved. While the switch is pressed, the tool moves at the feedrate specified in the parameter No. 1423. The tool stops when the switch is released.
3 The jog feedrate can be adjusted with the jog feedrate override dial.
4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate while the rapid traverse switch is pressed. Rapid traverse override by the rapid traverse override switches is effective during rapid traverse.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanation

- Manual per revolution feed

The manual per revolution feed is enabled for jog feed by setting bit 4 (JRV) of parameter No.1402.
During the manual per revolution feed, the tool is jogged at the feedrate that is obtained by multiplying the spindle speed by the feed amount per revolution, which is calculated by multiplying the jog feedrate override value by the feed amount per revolution specified with the parameter No. 1423.
During manual per revolution feed, the tool is jogged at the following feedrate:
Feed distance per rotation of the spindle (mm/rev) (specified with parameter No. 1423) × JOG feedrate override × actual spindle speed (rev/min).

Limitation

- Acceleration/deceleration for rapid traverse

Feedrate, time constant and method of automatic acceleration/ deceleration for manual rapid traverse are the same as G00 in programmed command.

- Change of modes

Changing the mode to the jog mode while pressing a feed axis and direction selection switch does not enable jog feed. To enable jog feed, enter the jog mode at first, then press a feed axis and direction selection switch.

- Rapid traverse prior to reference position return

If reference position return is not performed after power-on, pushing rapid traverse button does not actuate the rapid traverse but the remains at the JOG feedrate. This function can be disabled by setting bit 0 (RPD) of parameter No.1401.

3.3 INCREMENTAL FEED

In the incremental (INC) mode, pressing a feed axis and direction selection switch on the machine operator's panel moves the tool one step along the selected axis in the selected direction. The minimum distance the tool is moved is the least input increment. Each step can be 10, 100, or 1000 times the least input increment.
With using bit 2 (HNT) of parameter No. 7103, each step can be additionally 10 times the lest input increment.
The feedrate set in parameter No. 1423 is applied.
By using the manual feedrate override signal, the feedrate can be increased or decreased.
The tool can also be moved at the rapid traverse rate by using the manual rapid traverse selection signal, independent of the manual feedrate override signal.

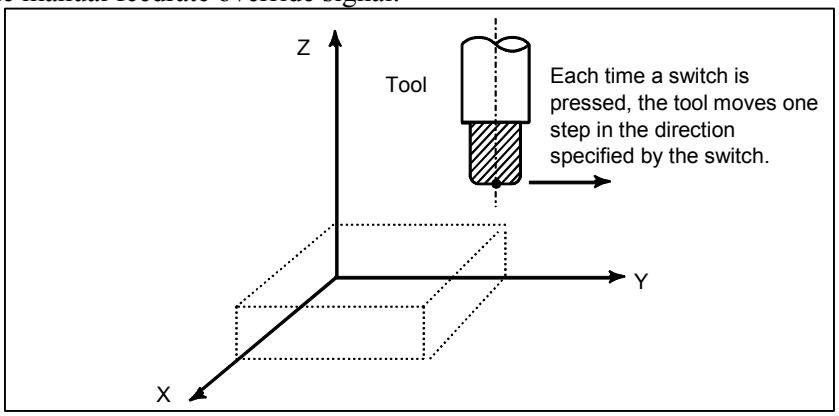


Fig. 3.3 (a) Incremental feed

Procedure for incremental feed

Procedure

1 Press the INC switch, one of the mode selection switches.
2 Select the distance to be moved for each step with the magnification dial.
3 Press the feed axis and direction selection switch corresponding to the axis and direction the tool is to be moved. Each time a switch is pressed, the tool moves one step. The feedrate is the same as the jog feedrate.
4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate.
Rapid traverse override by the rapid traverse override switch is effective during rapid traverse.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanation

- Travel distance specified with a diameter

T

The distance the tool travels along the X-axis can be specified with a diameter.

3.4 MANUAL HANDLE FEED

In the handle mode, the tool can be minutely moved by rotating the manual pulse generator on the machine operator's panel. Select the axis along which the tool is to be moved with the handle feed axis selection switches.
The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment. One of four types of magnifiers selected with the manual handle feed move distance select signal can be applied. With bit 2 (HNT) of parameter No. 7103, the minimum distance can be further 10 times greater.
The number of manual pulse generators depends on the option as follows:

- Control on 1 manual handle: Up to 1
- Control on 2 or 3 manual handles:
  Up to 3 (Up to three manual pulse generators can be moved at a time.)
- Control on 2 to 5 manual handles:
  Up to 5 (Up to five manual pulse generators can be moved at a time.)

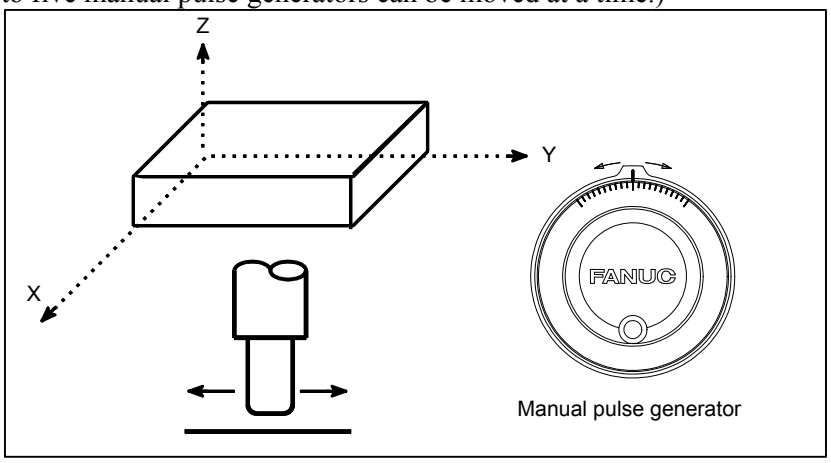


Fig. 3.4 (a) Manual handle feed

Procedure for manual handle feed

Procedure

1 Press the handle switch, one of the mode selection switches.
2 Select the axis along which the tool is to be moved by pressing a handle feed axis selection switch.
3 Select the magnification for the distance the tool is to be moved by pressing a handle feed magnification switch. The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment.
4 Move the tool along the selected axis by rotating the handle. Rotating the handle 360 degrees moves the tool the distance equivalent to 100 graduations.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanation

- Availability of manual pulse generator in Jog mode (JHD)

Bit 0 (JHD) of parameter No.7100 enables or disables the manual handle feed in the JOG mode.
When bit 0 (JHD) of parameter No.7100 is set 1,both manual handle feed and incremental feed are enabled.

- Manual handle feed in TEACH IN JOG mode (THD)

By setting bit 1 (THD) of parameter No. 7100, manual handle feed in TEACH IN JOG mode can be enabled or disabled.

- When manual handle feed exceeding the rapid traverse rate is specified

The amount of pulses exceeding the rapid traverse rate can be saved by CNC as B. And amount of pulses B will be output as pulses C.

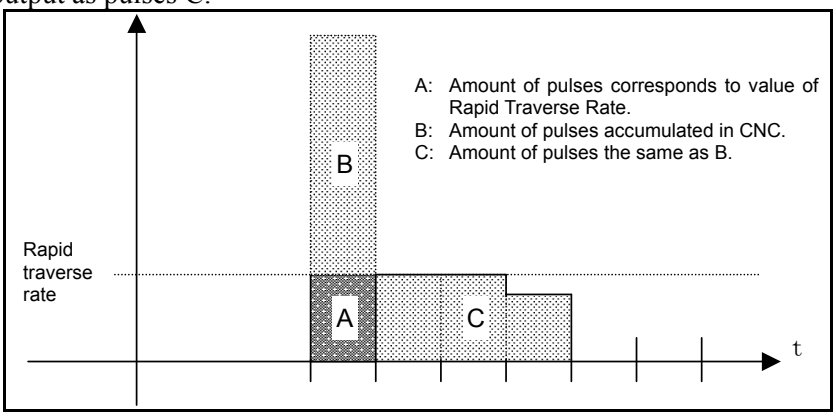


Fig. 3.4 (b) Amount of pulses output by CNC in Manual Handle Feed

Amount of pulses B is calculated in 2 cases as following:

1) In case of parameter No.7117 = 0
   The feedrate is clamped at the Rapid Traverse Rate and generated pulses exceeding the Rapid Traverse Rate are ignored (B=0).

2) In case of parameter No.7117 > 0
   The feedrate is clamped as the Rapid Traverse Rate, but the pulses exceeding the Rapid Traverse Rate is not ignored. Amount of pulses accumulated in CNC is calculated as following. (Although the rotation of manual pulse generator is stopped, if there is pulses accumulated in CNC, it will be output and the tool will move as long as amount of it.)

   Magnification set by manual handle feed selection signals MP1 and MP2 is m, value of parameter No.7117 is n.

   n < m: Clamping is set performed at value of parameter No.7117.
   n ≥ m: Amount A+B, shown in Fig. 3.4 (c), which's value is multiple of m and small than n. As a result, clamping is performed as an integral multiple of the selected magnification.

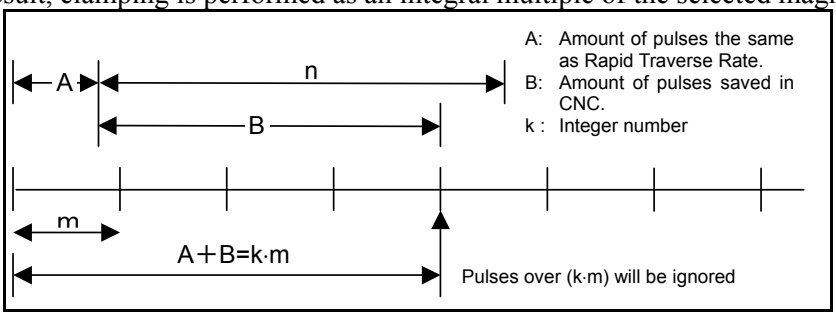


Fig. 3.4 (c) Amount of pulses exceeding the Rapid Traverse Rate (n ≥ m)

NOTE
Due to change of mode, clamping can be performed not as an integral multiple of the selected magnification.
The distance the tool moves may not match the graduations on the manual pulse generator.

- Upper feedrate limit in manual handle feed

The upper feedrate limit depends on the input signal (maximum manual handle feedrate switch signal HNDLF) from the PMC as follows:

- When HNDLF is set to 0, the feedrate is clamped to the manual rapid traverse rate (parameter No. 1424).
- When HNDLF is set to 1, the feedrate is clamped to the feedrate set in parameter No. 1434.

- Movement direction of an axis to the rotation of MPG (HNGx)

Bit 0 (HNGx) of parameter No 7102 switches the direction of MPG in which the tool moves along an axis, corresponding to the direction in which the handle of the manual pulse generator is rotated.

This parameter is valid only for the following functions:

- Manual handle feed
- Manual handle interruption

Limitation

⚠ WARNING

Rotating the handle quickly with a large magnification such as ×100 moves the tool too fast. The feedrate is clamped at the rapid traverse feedrate.

NOTE

Rotate the manual pulse generator at a rate of five rotations per second or lower. If the manual pulse generator is rotated at a rate higher than five rotations per second, the tool may not stop immediately after the handle is no longer rotated or the distance the tool moves may not match the graduations on the manual pulse generator.

3.5 MANUAL ABSOLUTE ON AND OFF

Whether the distance the tool is moved by manual operation is added to the coordinates can be selected by turning the manual absolute switch on or off on the machine operator's panel. When the switch is turned on, the distance the tool is moved by manual operation is added to the coordinates. When the switch is turned off, the distance the tool is moved by manual operation is not added to the coordinates.

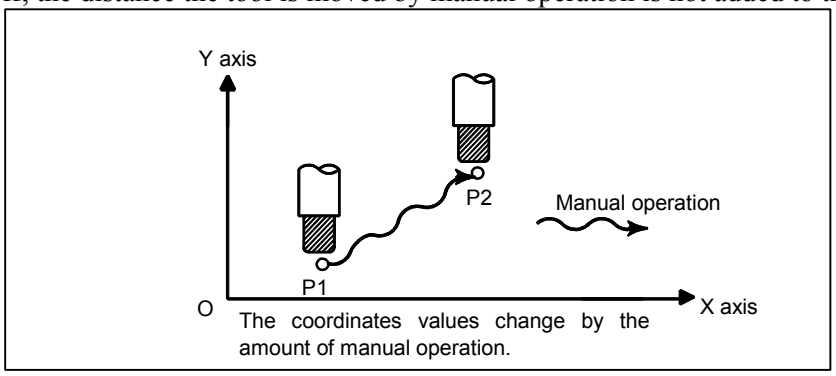


Fig. 3.5 (a) Coordinates with the switch ON

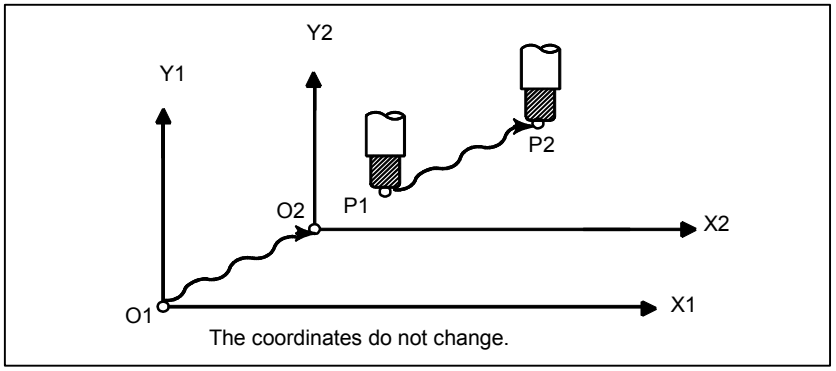


Fig. 3.5 (b) Coordinates with the switch OFF

Explanation

The following describes the relation between manual operation and coordinates when the manual absolute switch is turned on or off, using a program example.

```
G01G90  X100.0Y100.0F010 ;      <1>
        X200.0Y150.0     ;      <2>
        X300.0Y200.0     ;      <3>
```

Fig. 3.5 (c) Program example

The subsequent figures use the following notation:

→ Movement of the tool when the switch is on

- - - → Movement of the tool when the switch is off

The coordinates after manual operation include the distance the tool is moved by the manual operation. When the switch is off, therefore, subtract the distance the tool is moved by the manual operation.

- Manual operation after the end of block

Coordinates when block <1> has been executed after manual operation (X-axis +20.0, Y-axis +100.0) at the end of movement of block <2>.

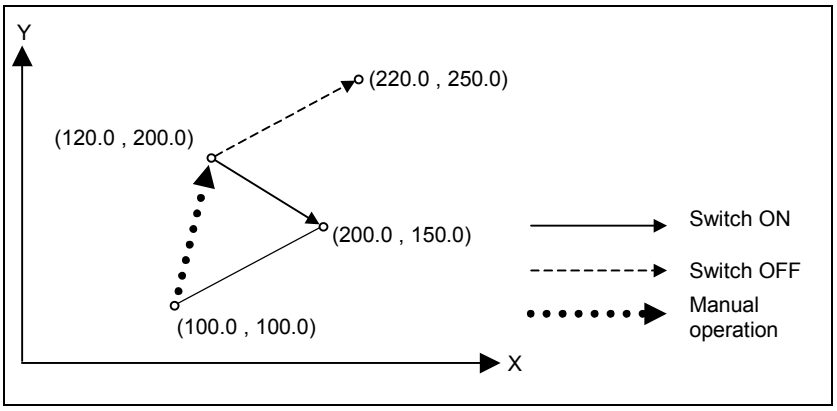


Fig. 3.5 (d) Manual operation after the end of block

- Manual operation after a feed hold

Coordinates when the feed hold button is pressed while block <2> is being executed, manual operation (Y-axis + 75.0) is performed, and the cycle start button is pressed and released.

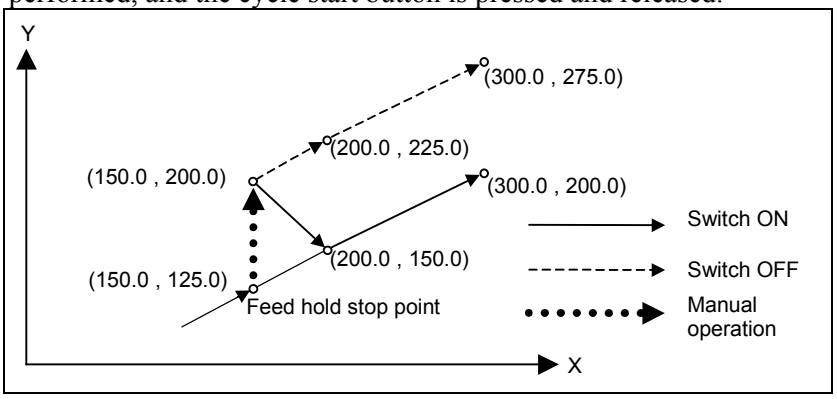


Fig. 3.5 (e) Manual operation after a feed hold

- When reset after a manual operation following a feed hold

Coordinates when the feed hold button is pressed while block <2> is being executed, manual operation (Y-axis +75.0) is performed, the control unit is reset with the RESET button, and block <2> is read again.

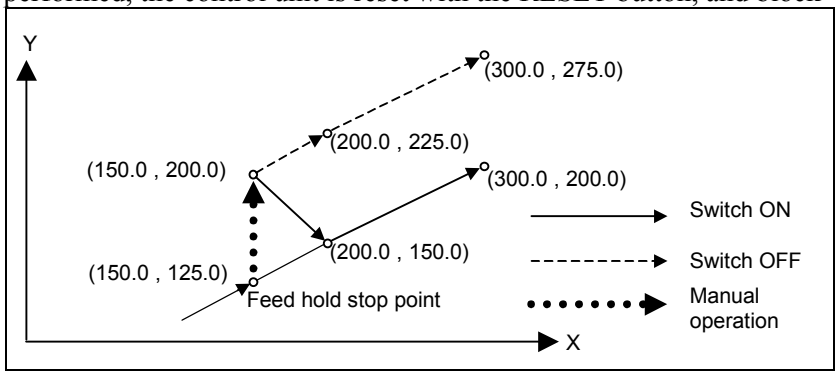


Fig. 3.5 (f) When reset after a manual operation following a feed hold

- When a movement command in the next block is only one axis

When there is only one axis in the following command, only the commanded axis returns.

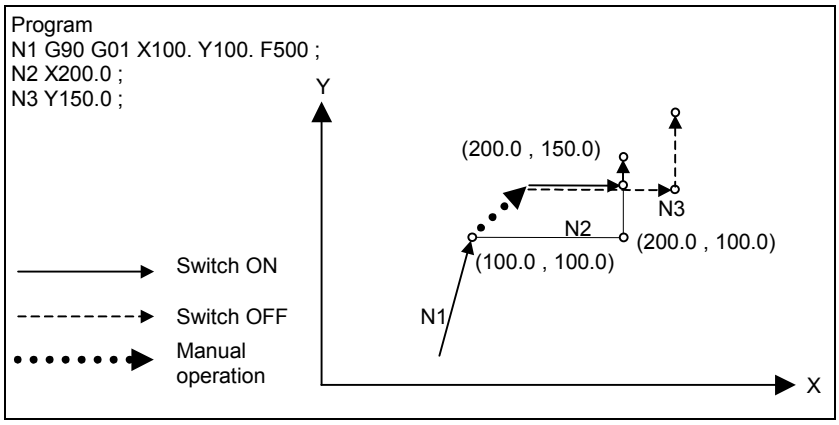


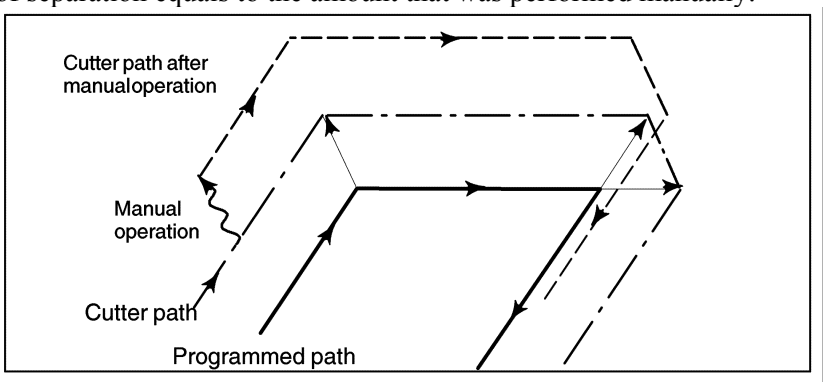
Fig. 3.5 (g) When a movement command in the next block is only one axis

- When the next move block is an incremental

When the following commands are incremental commands, operation is the same as when the switch is OFF.

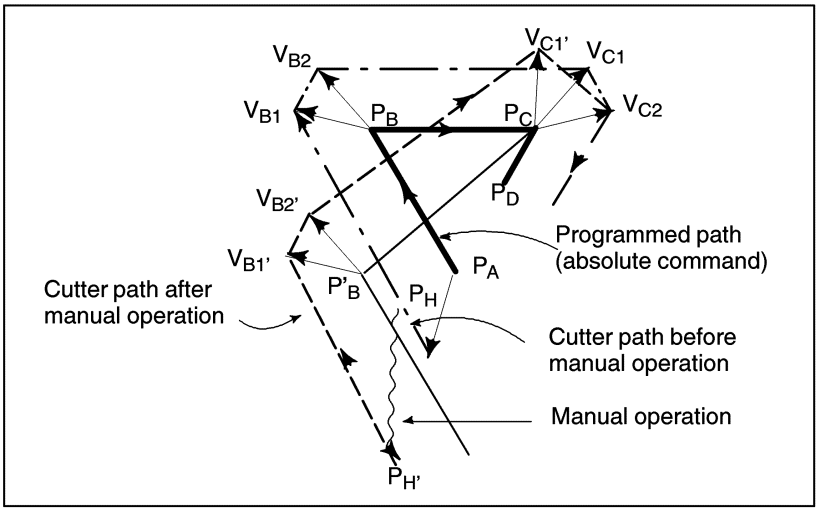
- Manual operation during cutter or tool nose radius compensation

- When the switch is OFF
  After manual operation is performed with the switch OFF during cutter or tool nose radius compensation, automatic operation is restarted then the tool moves parallel to the movement that would have been performed if manual movement had not been performed.
  The amount of separation equals to the amount that was performed manually.

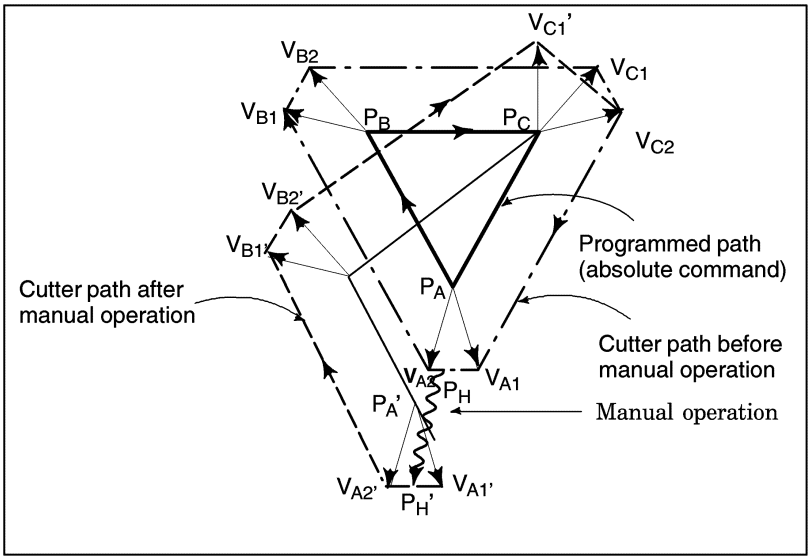


- When the switch is ON during cutter or tool nose radius compensation
  Operation of the machine upon return to automatic operation after manual intervention with the switch is ON during execution with an absolute command program in the cutter or tool nose radius compensation mode will be described. The vector created from the remaining part of the current block and the beginning of the next block is shifted in parallel. A new vector is created based on the next block, the block following the next block and the amount of manual movement. This also applies when manual operation is performed during cornering.

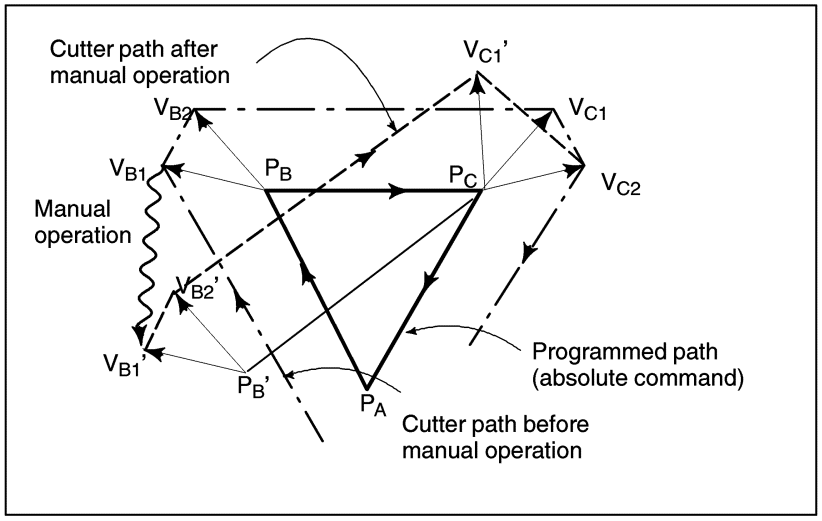
- Manual operation performed in other than cornering
  Assume that the feed hold was applied at point P_H while moving from P_A to P_B of programmed path P_A, P_B, and P_C and that the tool was manually moved to PH'. The block end point P_B moves to the point P_B' by the amount of manual movement, and vectors V_{B1} and V_{B2} at P_B also move to V_{B1}' and V_{B2}'. Vectors V_{C1} and V_{C2} between the next two blocks P_B - P_C and P_C - P_D are discarded and new vectors V_{C1}' and V_{C2}' (V_{C2}' = V_{C2} in this example) are produced from the relation between P_B' - P_C and P_C - P_D. However, since V_{B2}' is not a newly calculated vector, correct offset is not performed at block P_B' - P_C. Offset is correctly performed after P_C.



- Manual operation during cornering
  This is an example when manual operation is performed during cornering. V_{A2}', V_{B1}', and V_{B2}' are vectors moved in parallel with V_{A2}, V_{B1} and V_{B2} by the amount of manual movement. The new vectors are calculated from V_{C1} and V_{C2}. Then correct cutter or tool nose radius compensation is performed for the blocks following P_C.



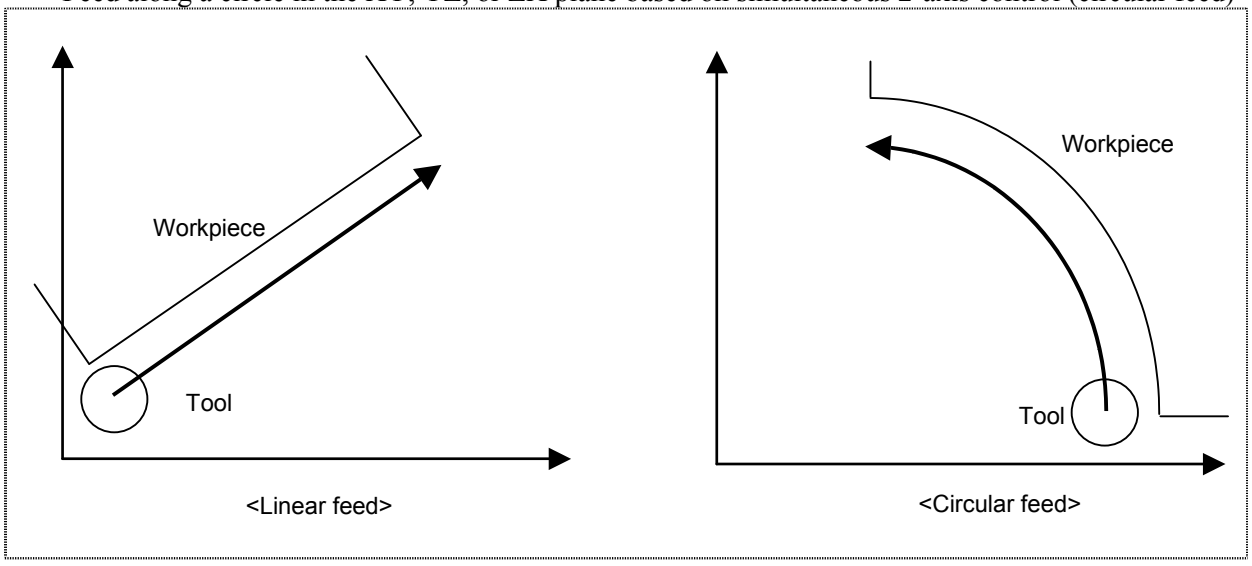
- Manual operation after single block stop
  Manual operation was performed when execution of a block was terminated by single block stop. Vectors V_{B1} and V_{B2} are shifted by the amount of manual operation. Sub-sequent processing is the same as case a described above. An MDI operation can also be intervened as well as manual operation. The movement is the same as that by manual operation.



3.6 MANUAL LINEAR/CIRCULAR INTERPOLATION

In manual handle feed or jog feed, the following types of feed operations are possible along with the conventional feed operation with simultaneous single-axis control (for X, Y, Z, or other axis).

- Feed along a tilted straight line in the XY, YZ, or ZX plane based on simultaneous 2-axis control (linear feed)
- Feed along a circle in the XY, YZ, or ZX plane based on simultaneous 2-axis control (circular feed)



NOTE

1 Two control axes should be included in the three standard axes.
2 In the following, only the case of the XY plane is described. If any other plane is used, replace XY with YZ or ZX. (Refer to the table below.)

XY plane	YZ plane	ZX plane
X	Y	Z
Y	Z	X

Explanations

Procedure

1 For manual handle feed, select the manual handle feed mode. For jog feed, select the jog feed mode.
2 For manual handle feed, use the handle feed axis selection switch to select the feed axis (simultaneous 1-axis feed in the X-, Y-, or Z-axis or simultaneous 2-axes feed of the X- and Y-axes along a specified line or circle) used when the manual handle is turned.
For jog feed, use the feed axis direction selection switch to select the feed axis and its direction. The tool moves at the feedrate (jog feedrate) set in parameter No. 1423 in the specified axis direction or along the specified line or circle while the feed axis and its direction are selected.
3 For the manual handle, the tool moves in the specified axis direction when the manual handle is turned. The feedrate depends on the rotation speed of the manual handle. The amount of movement when the manual handle is turned for one pulse can be selected by the manual handle feed movement scale switch.
For jog feed, the manual feedrate override dial can be used to override the feedrate.

The above procedure is an example. For actual operation, refer to the manual provided by the machine tool builder.

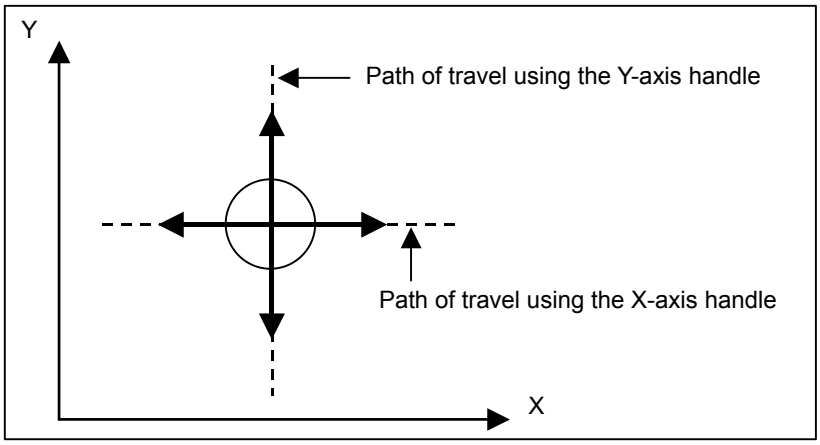
- Line/circle definition

A line or circle definition is not required for axis direction feed. For linear or circular feed, however, a line or circle must be defined. (For example, in the case of circular feed, the center and radius of the circle must be specified.)
Refer to the manual provided by the machine tool builder.

- Manual handle feed

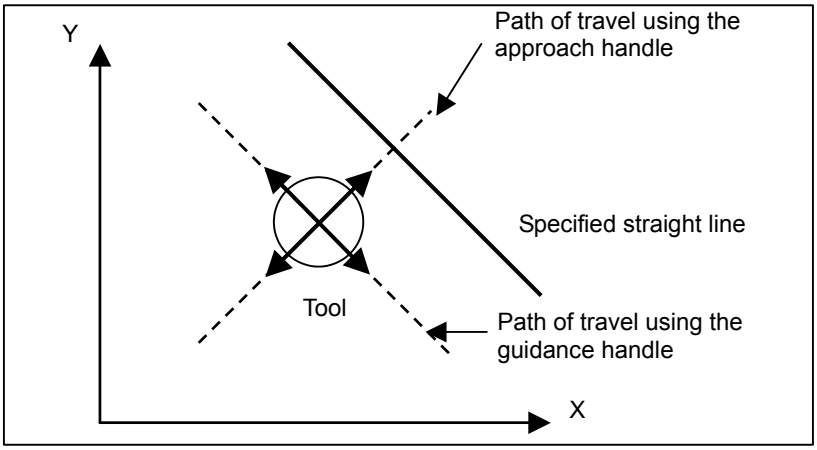
In manual handle feed, the tool can be moved along a specified axis (X-axis, Y-axis, Z-axis, or N^{th}-axis), along a rotated straight line (linear feed), or along a circle (circular feed).

(1) Feed along a specified axis (simultaneous 1-axis control)
A single manual handle operation can move the tool along the desired axis, such as X-axis, Y-axis, or Z-axis, on a simultaneous 1-axis control basis.



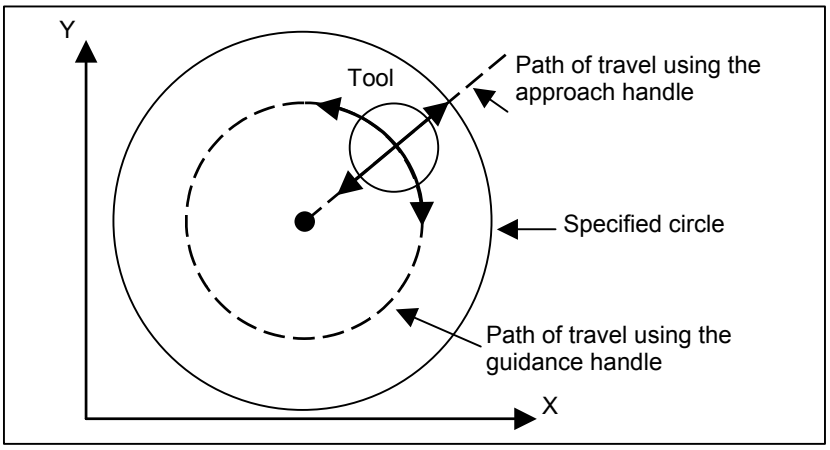
Feed along a specified axis

(2) Linear feed (simultaneous 2-axis control)
A single manual handle operation can move the tool in parallel to the straight line on a simultaneous 2-axis control basis. This manual handle is referred to as the "guidance handle". Also, the tool can be moved at right angles to a specified straight line on a simultaneous 2-axis control basis. This manual handle is referred to as the "approach handle". When the guidance handle or approach handle is turned clockwise or counterclockwise, the tool travels forward or backward along the respective path.



Linear feed

(3) Circular feed (simultaneous 2-axis control)
A single manual handle operation can move the tool from the current position along a concentric circle that has the same center as a specified circle on a simultaneous 2-axis control basis. This manual handle is referred to as the guidance handle. Also, a single manual handle operation can move the tool along a straight line normal to a specified circle on simultaneous 2-axis control. This manual handle is referred to as the approach handle. When the guidance handle or approach handle is turned clockwise or counterclockwise, the tool travels forward or backward along the respective path.



Circular feed

- Feedrate for manual handle feed

The feedrate depends on the speed at which a manual handle is turned. The distance to be traveled by the tool (along a tangent in the case of linear or circular feed) per manual handle pulse can be selected using the manual handle feed travel distance magnification switch.
The approach handle clamp feedrate can be set using parameter No. 7160. The guidance handle clamp feedrate can be set using parameter No. 7161. Set the parameter No. 7117 to 0.

- Manual handle selection

Up to five manual handles to be connected. For information on specifying the usage of the manual handle connected to each interface (for feed in the axis direction, use as a guidance handle, or use as an approach handle), refer to the manual issued by the machine tool builder.

- Direction of movement using manual handles

When the tool is moved along a straight line or circle using the guidance handle or approach handle, the direction of the tool movement (for example, whether to make a clockwise or counterclockwise movement along a circle) when the handle is turned clockwise or counterclockwise can be specified.

Refer to the manual provided by the machine tool builder.

- Jog feed

In jog feed, the tool can be moved along a specified axis (X-axis, Y-axis, Z-axis, etc.), along a rotated straight line (linear feed), or along a circle (circular feed).

(1) Feed along a specified axis (simultaneous 1-axis control)
While a feed axis and its direction are specified with the feed axis direction selection switch, the tool moves in the specified axis direction at the feedrate specified in parameter No. 1423. The feedrate can be overridden using the manual feedrate override dial. Manual rapid traverse selection signal (RT) is invalid.

(2) Linear feed (simultaneous 2-axis control)
By defining a straight line, the tool can be moved as follows:
- While a feed axis and its direction are selected using the feed axis direction selection switch, the tool moves along a straight line parallel to the specified straight line on a simultaneous 2-axis control basis.
- While a feed axis and its direction are selected using the feed axis direction selection switch, the tool moves along a straight line perpendicular to the specified straight line on a simultaneous 2-axis control basis.

The feedrate in the tangential direction is specified in parameter No. 1410.
The feedrate can be overridden using the manual feedrate override dial. Manual rapid traverse selection signal (RT) is invalid.

(3) Circular feed (simultaneous 2-axis control)
By defining a circle, the tool can be moved as follows:
- While a feed axis and its direction are selected using the feed axis direction selection switch, the tool moves from the current position along the concentric circle that has the same center as the specified circle.
- While a feed axis and its direction are selected using the feed axis direction selection switch, the tool moves along a straight line normal to the specified circle.

The feedrate in the tangential direction is specified in parameter No. 1410. The feedrate can be overridden using the manual feedrate override dial. Manual rapid traverse selection signal (RT) is invalid.

- Manual handle feed in JOG mode

Even in JOG mode, manual handle feed can be enabled using bit 0 (JHD) of parameter No. 7100. In this case, however, manual handle feed is enabled only when the tool is not moved using jog feed.

Limitations

- Mirror image

The mirror image function is not available during the manual operation. (The manual operation can be executed when the mirror image switch is off and the mirror image setting is off.)

NOTE
If the tool is operated with the mirror image function activated, the normal operation will not be guaranteed.

- Rotation direction

When this function is effective, bit 0 (HNGx) of parameter No. 7102 is invalid which is used to change the axis movement direction with respect to the rotation direction of the manual pulse generator. (The rotation direction of the manual pulse generator is always the same as the axis movement direction; that is, the operation is the same as when bit 0 of parameter No. 7102 is set to "0".)

- The amount of the shift for manual handle feed

When this function is effective, parameters Nos. 12350 and 12351 used to determine the magnification of manual handle feed for each axis are invalid and the values of parameters Nos. 7113 and 7114 are used. (Parameters No. 7131 and No. 7132 are used for the second manual pulse generator, parameters Nos. 7133 and 7134 for the third, parameters Nos. 7135 and 7136 for the fourth, and parameters Nos. 7137 and 7138 for fifth.)

- Incremental feed

This function is invalid for the incremental feed.

- Synchronous/composite control

If the axis is set as the synchronous/composite control axis, this function is invalid.

3.7 RIGID TAPPING BY MANUAL HANDLE

For execution of rigid tapping, set rigid mode, then switch to handle mode and move the tapping axis with a manual handle. For rigid tapping, refer to Section, “RIGID TAPPING” in the OPERATOR’S MANUAL (T series) or Section, “RIGID TAPPING” in the OPERATOR’S MANUAL (M series) and the relevant manual of the machine tool builder.

Rigid tapping by manual handle

Procedure

1 Stop the spindle and servo axes, then set MDI mode by pressing the MDI switch among the mode selection switches.

2 Enter and execute the following program:

Example 1) M series / T series (G code systems B, C)

```
M29 S100 ;
G91 G84 Z0 F1000 ;
```

Example 2) T series (G code system A)

```
M29 S100 ;
G84 W0 F1000 ;
```

The program above is required to determine a screw lead and set rigid tapping mode. In this program, a tapping axis must always be specified. Specify a value that does not operate the tapping axis. With the M series, specify G84 (G74). With the T series, specify G84 (G88).

> ⚠ **WARNING**
> In this MDI programming, never specify commands to position the tool at a drilling position and at point R. Otherwise, the tool moves along an axis.

3 When the entered program is executed, rigid tapping mode is set.

4 After rigid mode is set upon completion of MDI program execution, switch to the handle mode by pressing the handle switch among the mode selection switches.

> ⚠ **CAUTION**
> At this time, never press the reset key. Otherwise, rigid mode is canceled.

5 To perform rigid tapping, select a tapping axis with the handle feed axis select switch, and move the tapping axis with the manual handle.

Explanation

- Manual rigid tapping

Manual rigid tapping is enabled by bit 0 (HRG) of parameter No. 5203 to 1.

- Cancellation of rigid mode

To cancel rigid mode, specify G80 as same the normal rigid tapping. When the reset key is pressed, rigid mode is canceled, but the canned cycle is not canceled.
When the rigid mode switch is to be set to off for rigid mode cancellation (when bit 2 (CRG) of parameter No. 5200 is set to 0), the G80 command ends after the rigid mode switch is set to off.

- Spindle rotation direction

The rotation direction of the spindle is determined by a specified tapping cycle G code and the setting of bit 1 (HRM) of parameter No. 5203. For example, when the HRM parameter is set to 0 in G84 mode, the spindle makes forward rotations as the tapping axis moves in the minus direction. (When the tapping axis moves in the plus direction, the spindle makes reverse rotations.)

- Arbitrary tapping axis

With the M series, an arbitrary tapping axis can be selected by setting bit 0 (FXY) of parameter No. 5101 to 1. With the T series, an arbitrary tapping axis can be selected not by using bit 0 (FXY) of parameter No. 5101 (but by using bit 0 (FXY) of parameter No. 5101 when the FS15 format is specified). In this case, specify a G code for plane selection and the address of the tapping axis when setting the rigid tapping mode in the MDI mode.

- G84 command/same block specifying both M29 and G84

As the MDI program command for setting the rigid mode, G84 can be specified as a rigid tapping G code (by setting bit 0 (G84) of parameter No. 5200 to 1), or M29 and G84 can be specified in the same block.
Example 1: G91 G84 Z0 F1000 S100 ;
Example 2: G91 G84 Z0 F1000 M29 S100 ;

- Specification of manual handle feed faster than the rapid traverse rate

Set parameter No.7117 to 0 so that when manual handle feed is specified which is faster than the rapid traverse rate, the handle pulses beyond the rapid traverse rate are ignored.

- Series 15 format command

By setting bit 1 (FCV) of parameter No. 0001 to 1, the Series15 format can be used for specification.
Example 1: G91 G84.2(G84.3) Z0 F1000 S100; (M series)
Example 2: G84.2 W0 F1000 S100; (T series : G code systems A)
Example 3: G91 G84.2 Z0 F1000 S100; (T series : G code system B, C)

- Acceleration/deceleration type

When manual rigid tapping is executed, the acceleration/deceleration type and acceleration/deceleration time constant set in the rigid tapping parameters are valid.
The same settings are valid also for extraction.

- In the case of multi-spindle operation

In the case of multi-spindle operation, a spindle can be selected by specifying a P command as well as an S command.
Example : When the 2nd spindle is selected:

```
M29 S100 P2;
G91 G84 Z0 F1000 ;
```

Limitation

- Excessive error check

In manual rigid tapping, only an excessive error during movement is checked.

- Tool axis direction handle feed

Tool axis direction handle feed is disabled.

- Extraction override

In manual rigid tapping, the extraction override function is disabled, and the use of an acceleration/deceleration time constant for extraction is disabled.

- Number of repeats

In MDI programming, never specify K0 and L0, which are used to specify that the number of repeats is 0 and to disable the execution of a G84 block. If K0 or L0 is specified, rigid mode cannot be set.

- Positioning of the tool to a drilling position

When positioning the tool to a drilling position, select the X-axis or Y-axis with the axis select switch in handle mode. Never use the method of positioning to a drilling position in MDI mode or MEM mode. The method can operate the tapping axis.

- 3-dimensional rigid tapping

3-dimensional rigid tapping cannot be used in rigid tapping using the manual handle.

- Interpolation type rigid tapping

Interpolation type rigid tapping cannot be used in rigid tapping using the manual handle. To perform rigid tapping using the manual handle, select the conventional type of rigid tapping with bit 3 (CHR) of parameter No. 5202.

- Feed forward

In rigid tapping by manual handle, feed forward is disabled even if bit 2 (RFF) of parameter No. 5203 is set to 1 (Feed forward is enabled in rigid tapping).

3.8 MANUAL NUMERICAL COMMAND

The manual numerical command function allows data programmed through the MDI to be executed in jog mode. Whenever the system is ready for jog feed, a manual numerical command can be executed. The following eight functions are supported:

(1) Positioning (G00)
(2) Linear interpolation (G01)
(3) Automatic reference position return (G28)
(4) 2nd/3rd/4th reference position return (G30)
(5) M codes (miscellaneous functions)
(6) S codes (spindle speed functions)
(7) T codes (tool functions)
(8) B codes (second auxiliary functions)

By setting the following parameters, the commands for axial motion and the M, S, T, and B functions can be disabled:

(1) Positioning (G00): ..... Bit 0 (JAXx) of parameter No. 7010
(2) Linear interpolation (G01): ..... Bit 0 (JAXx) of parameter No. 7010
(3) Automatic reference position return (G28): ..... Bit 0 (JAXx) of parameter No. 7010
(4) 2nd/3rd/4th reference position return (G30): ..... Bit 0 (JAXx) of parameter No. 7010
(5) M codes (miscellaneous functions): ..... Bit 0 (JMF) of parameter No. 7002
(6) S codes (spindle speed functions): ..... Bit 1 (JSF) of parameter No. 7002

(7) T codes (tool functions): .....Bit 2 (JTF) of parameter No. 7002
(8) B codes (second auxiliary functions): .....Bit 3 (JBF) of parameter No. 7002

ProcedureManual numerical command

Procedure

1 Press the jog switch (one of the mode selection switches).
2 Press function key 

.
3 Press soft key [JOG] on the screen. The following manual numerical command screen is displayed.

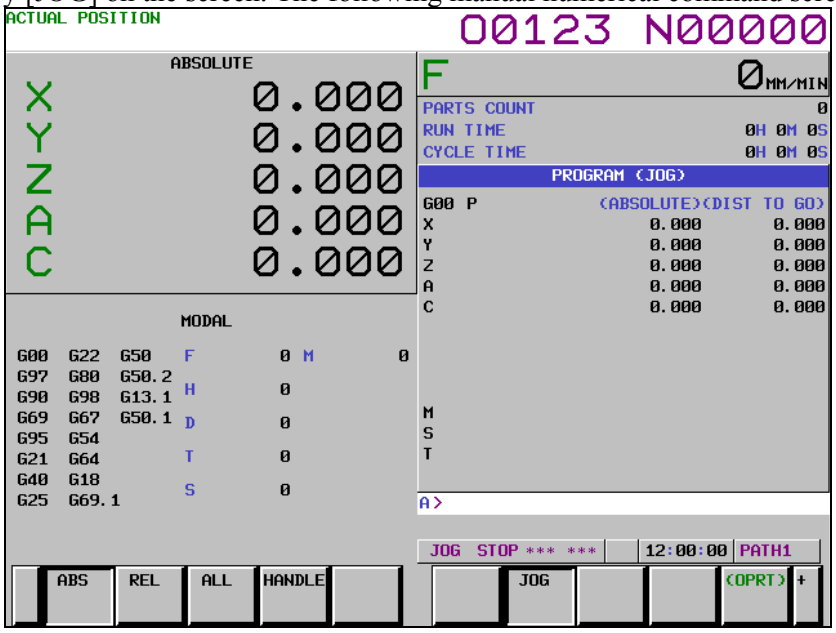


Fig. 3.8 (a) Manual numerical command screen

The remaining portion of the axis information currently not shown on the screen can be displayed by pressing the 

or 

key.

NOTE
1 The actual feedrate (F) and the actual spindle speed (S) are displayed only for 8.4-inch window.
2 If two or more identical axis names exist for the same path, they are not displayed.

4 Enter the required commands by using address keys and numeric keys on the MDI unit, then press soft key [INPUT] or the  key to set the entered data.

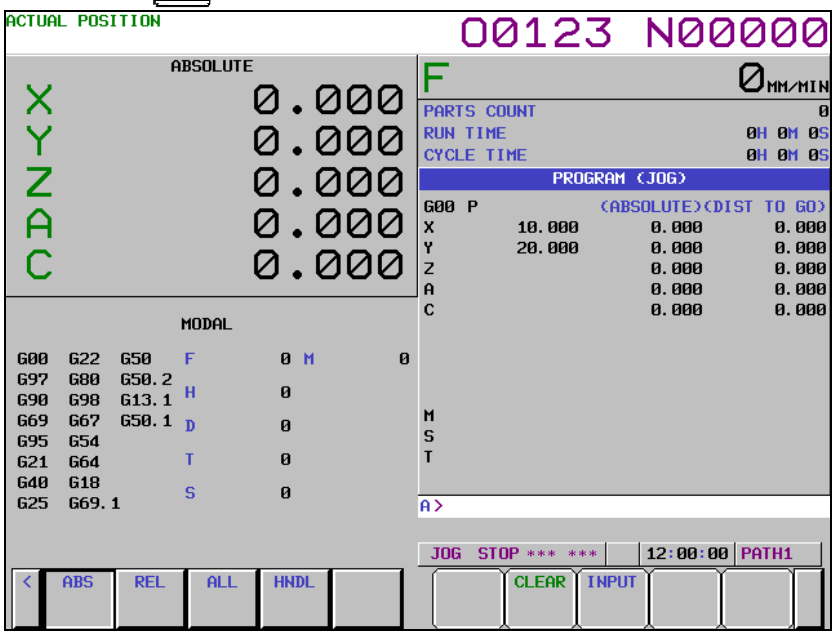


Fig. 3.8 (b) Example of inputting numerical value

The following data can be set:

1. G00: Positioning
2. G01: Linear interpolation
3. G28: Automatic reference position return
4. G30: 2nd/3rd/4th reference position return
5. M codes: Miscellaneous functions
6. S codes: Spindle speed functions
7. T codes: Tool functions
8. B codes: Second auxiliary functions

The set data is maintained even when the screen or mode is changed.

NOTE
When an alarm state exists, data cannot be set.

5 Press the cycle start switch on the machine operator's panel to start command execution. The status is indicated as "MSTR." The automatic operation signal, STL, can be turned on by setting bit 2 (JST) of parameter No.7001.

NOTE
If the cycle start switch is pressed while an alarm state exists, a "START REJECTED (ALARM EXISTED)" warning is generated, and the entered data cannot be executed.

6 Upon the completion of execution, the "MSTR" status indication is cleared from the screen, and automatic operation signal STL is turned off. The set data is cleared entirely. G codes are set to G00 or G01 according to the setting of bit 0 (G01) of parameter No. 3402.

Explanation

- Positioning

An amount of travel is given as a numeric value, preceded by an address such as X, Y, or Z. This is always regarded as being an incremental command, regardless of whether G90 or G91 is specified.

	Manual rapid traverse selection switch	
	Off	**On**
Feedrate (parameter)	Jog feedrate for each axis (No.1423)	Rapid traverse rate for each axis (No.1420)
Automatic acceleration/ deceleration (parameter)	Exponential acceleration/ deceleration in jog feed for each axis (No.1624)	Linear acceleration/deceleration in rapid traverse for each axis (No.1620)
Override	Manual feed override	Rapid traverse override

NOTE
1 When the manual rapid traverse switch is off, the feedrate is clamped so that the axis-by-axis manual rapid traverse rate (parameter No. 1424) is not exceeded.
2 The tool cannot move if linear interpolation type positioning (where the tool path is linear) has been performed by setting bit 1 (LRP) of parameter No. 1401.

- Linear interpolation (G01)

An amount of travel is given as a numeric value, preceded by an address such as X, Y, or Z. This is always regarded as being an incremental command, regardless of whether G90 or G91 is specified. Axial movements are always performed in incremental mode even during scaling or polar coordinate interpolation. In addition, movement is always performed in feed per minute mode regardless of the specification of G94 or G95.

Feedrate (parameter)	Dry run feedrate (No. 1410)
Automatic acceleration/deceleration (parameter)	Exponential acceleration/deceleration in cutting feed for each axis (No.1622)
Override	Manual feed override

NOTE
Since the feedrate is always set to the dry run feedrate, regardless of the setting of the dry run switch, the feedrate cannot be specified using F. The feedrate is clamped such that the maximum cutting feedrate, set in parameter No. 1430, is not exceeded.

- Automatic reference position return (G28)

The tool returns directly to the reference position without passing through any intermediate points, regardless of the specified amount of travel. For axes for which no move command is specified, however, a return operation is not performed.

Feedrate (parameter)	Rapid traverse rate (No. 1420)
Automatic acceleration/deceleration (parameter)	Linear acceleration/deceleration in rapid traverse for each axis (No. 1620)
Override	Rapid traverse override

- 2nd, 3rd, or 4th reference position return (G30)

The tool returns directly to the 2nd, 3rd, or 4th reference position without passing through any intermediate points, regardless of the specified amount of travel. To select a reference position, specify 2, 3, or 4 in address P. If address P is omitted, a return to the second reference position is performed.

Feedrate (parameter)	Rapid traverse rate (No. 1420)
Automatic acceleration/deceleration (parameter)	Linear acceleration/deceleration in rapid traverse for each axis (No. 1620)
Override	Rapid traverse override

NOTE
The function for 3rd/4th reference position return is optional.
1 When the option is not selected
If "P2" is not specified in address P, the alarm PS0046 is generated and the function cannot be executed.
2 When the option is selected
If "P2," "P3," or "P4" is not specified in address P, the alarm PS0046, “ILLEGAL REFERENCE RETURN COMMAND” is generated and the function cannot be executed.

- M codes (Auxiliary functions)

After address M, specify a numeric value of no more than the number of digits specified by parameter No. 3030. When M98 or M99 is specified, it is executed but not output to the PMC.

NOTE
Neither subprogram calls nor custom macro calls can be performed using M codes.

- S codes (spindle speed functions)

After address S, specify a numeric value of no more than the number of digits specified by parameter No. 3031.

NOTE
Subprogram calls cannot be performed using S codes.

- T codes (tool functions)

After address T, specify a numeric value of no more than the number of digits specified by parameter No. 3032.

NOTE
1 Subprogram calls cannot be performed using T codes.
2 In lathe systems, tool compensation by T code commands is not performed.

- B codes (second auxiliary functions)

After address B, specify a numeric value of no more than the number of digits specified by parameter No. 3033.

NOTE
1 B codes can be renamed "U," "V," "W," "A," or "C" by setting parameter No. 3460. If the new name is the same as an axis name address, "B" is used. Note that "U," "V," and "W" can be used for the T codes only when the G codes are B or C.
2 Subprogram calls cannot be performed using B codes.

- Data input

(1) When addresses and numeric values of a command are typed, then soft key [INPUT] is pressed, the entered data is set. In this case, the input unit is either the least input increment or calculator-type input format, according to the setting of bit 0 (DPI) of parameter No. 3401.

The  key on the MDI unit can be used instead of soft key [INPUT].

(2) Commands can be typed successively.

(3) Key entry is disabled during execution.

If soft key [INPUT] or the  key on the MDI unit is pressed during execution, an "EXECUTION/MODE SWITCHING IN PROGRESS" warning is output.

(4) If input data contains an error, the following warnings may appear:

Warning	Description
FORMAT ERROR	- A G code other than G00, G01, and G28 has been entered. - An address other than those displayed on the manual numerical command screen has been entered. A value that exceeds the following limitations has been entered. - Address G: .....2 digits - Address P: .....1 digit - Axis address:.....9 digits - M, S, T, B: .....The parameter-set number of digits

NOTE
Even when the memory protection key is set, key input can nevertheless be performed.

- Erasing data

(1) When soft key [CLEAR] is pressed, followed by soft key [EXEC], all the set data is cleared. In this case, however, the G codes are set to G00 or G01, depending on the setting of bit 0 (G01) of parameter No. 3402.

Data can also be cleared by pressing the  key on the MDI unit.

(2) If soft key [CLEAR] is pressed during execution, an "EXECUTION/MODE SWITCHING IN PROGRESS" warning is output.

- Scrolling pages

If the whole axis information cannot be displayed on a single page, you can scroll pages by pressing the  or  key.

- Halting execution

If one of the following occurs during execution, execution is halted, and the data is cleared in the same way as when soft key [CLEAR] is pressed. The remaining distance to be traveled is canceled.

(1) When a feed hold is applied
(2) When the mode is changed to other than jog feed mode
(3) When an alarm is generated
(4) When a reset or emergency stop is applied

The M, S, T, and B functions remain effective even upon the occurrence of the above events, with the exception of (4).

- Modal information

Modal G codes and addresses used in automatic operation or MDI operation are not affected by the execution of commands specified using the manual numerical command function.

- Jog feed

When the tool is moved along an axis using a feed axis and direction selection switch on the manual numerical command screen, the remaining amount of travel is always shown as "0".

- Disabling the M, S, T, and B functions

By setting bit 0 to bit 3 (JMF, JSF, JTF, and JBF) of parameter No. 7002, the M, S, T, and B functions can be disabled. If a disabled function is specified, the warning "THIS COMMAND CANNOT BE EXECUTED." is issued.

Limitation

- M, S, T, and B functions

While automatic operation is halted, manual numerical commands can be executed. In the following cases, however, a "START REJECTED (ALREADY EXECUTING)" warning is output, and command execution is disabled.

(1) When an M, S, T, or B function is already being executed, a manual numerical command containing an M, S, T, or B function cannot be executed.
(2) When an M, S, T, or B function is already being executed, and that function alone is specified or a block specifying that function also contains another function (such as a move command or dwell function) which has already been completed, a manual numerical command cannot be executed.

- Jog feed

When a manual numerical command is specified while the tool is being moved along an axis by using a feed axis and direction selection switch, the axial movement is interrupted, and the manual numerical command is executed. Therefore, the tool cannot be moved along an axis by using a feed axis and direction selection switch during execution of a manual numerical command.

- Mirror image

A mirror image cannot be produced for the direction of a specified axial movement.

- REF mode

The manual numerical command screen is not displayed in the REF mode.

- Indexing of the index table and chopping

Commands cannot be specified for an axis along which operation is being performed during indexing or chopping.
If such an axis is specified for execution, a "THIS COMMAND CAN NOT EXECUTE" warning is output.

- Functions that cannot be specified

Commands cannot be specified for axes that operate using any of the following functions.

- Index table indexing
- Chopping
- Spindle positioning
- Polygon turning
- Synchronous, composite, and superimposed control

If the commands are executed for any such axis, a "THIS COMMAND CAN NOT EXECUTE" warning is generated.

- Functions that cannot be used

Commands cannot be specified for the following functions.

- Extended axis name
- Extended spindle name

- Address P command for multi spindle
- Cs contour control function

3.9 3-DIMENSIONAL MANUAL FEED

This function enables the use of the following functions.

- 3-dimensional manual feed
  - Tool axis direction handle feed/tool axis direction JOG feed/tool axis direction incremental feed
  - Tool axis right-angle direction handle feed/tool axis right-angle direction JOG feed/tool axis right-angle direction incremental feed
  - Tool tip center rotation handle feed/tool tip center rotation JOG feed/tool tip center rotation incremental feed
  - Table vertical direction handle feed/table vertical direction JOG feed/table vertical direction incremental feed
  - Table horizontal direction handle feed/table horizontal direction JOG feed/table horizontal direction incremental feed

A handle interrupt can be generated for each handle feed. Handle interrupts work according to the corresponding handle feed specifications described hereinafter unless otherwise noted.

- Screen display functions
  - Display of the coordinate of the tool tip
  - Display of pulse values
  - Display of the amount of machine axes movement

- Selecting a coordinate system when calculating the tool direction

If, in 3-dimensional manual feed, a workpiece coordinate system offset is set for a rotation axis, use bit 0 (CAC) of parameter No. 12319 to select whether to use values in the workpiece or machine coordinate system when calculating the tool direction.

- When the bit 0 (CAC) of parameter No. 12319 = 0:
  Values in the machine coordinate system are used.
  For those parameters for configuring the machine, Nos. 19680 to 19714, that depend on the coordinate on a rotation axis, set the values assumed when the machine coordinate on the rotation axis is 0.

- When the bit 0 (CAC) of parameter No. 12319 = 1:
  Values in the workpiece coordinate system are used.
  For those parameters for configuring the machine, Nos. 19680 to 19714, that depend on the coordinate on a rotation axis, set the values assumed when the workpiece coordinate on the rotation axis is 0.

For the functions below, values in the workpiece coordinate system of a rotation axis are used to calculate the tool direction and, therefore, if a workpiece coordinate system offset is set for a rotation axis, and 3-dimensional manual feed is to be used together with any of the functions below, set bit 0 (CAC) of parameter No. 12319 to 1.

- Tool center point control
- Tilted working plane command
- Tool length compensation in tool axis direction

(Example)

Related parameters:
No.19680=2 (Tool rotation type)
No.19682=3 (the master rotation axis (C-axis) is about the Z-axis)

No.19687=2 (the slave rotation axis (B-axis) is about the Y-axis)
No.19697=3 (the reference tool axis direction is the Z-axis direction)
No.19698=0 (angle RA when the reference tool axis direction is tilted)
No.19699=0 (angle RB when the reference tool axis direction is tilted)

Workpiece coordinate system offset:
B=10.0

Example 1:

The tool is assumed to face the Z-axis direction when it is in the following states.
Position in the workpiece coordinate system B= -10.0
Position in the machine coordinate system B=0.0
In this case, values in the machine coordinate system must be used to calculate the tool direction, set bit 0 (CAC) of parameter No. 12319 to 0.

Example 2:

The tool is assumed to face the Z-axis direction when it is in the following states.
Position in the workpiece coordinate system B=0.0
Position in the machine coordinate system B=10.0
In this case, values in the workpiece coordinate system must be used to calculate the tool direction, set bit 0 (CAC) of parameter No. 12319 to 1.

NOTE
1 To execute 3-dimensional handle feed requires the manual handle feed option. Also, to generate 3-dimensional handle interrupts requires the manual handle interrupt option.
2 A 3-dimensional handle interrupt must not be generated when a rotation axis command is being executed during automatic operation.
3 3-dimensional manual feed is disabled when the manual reference position return mode is selected.
4 If per-axis interlock is enabled to at least one of 3-dimensional manual feed axes, movement with manual feed is not performed.

3.9.1 Tool Axis Direction Handle Feed / Tool Axis Direction JOG Feed / Tool Axis Direction Incremental Feed

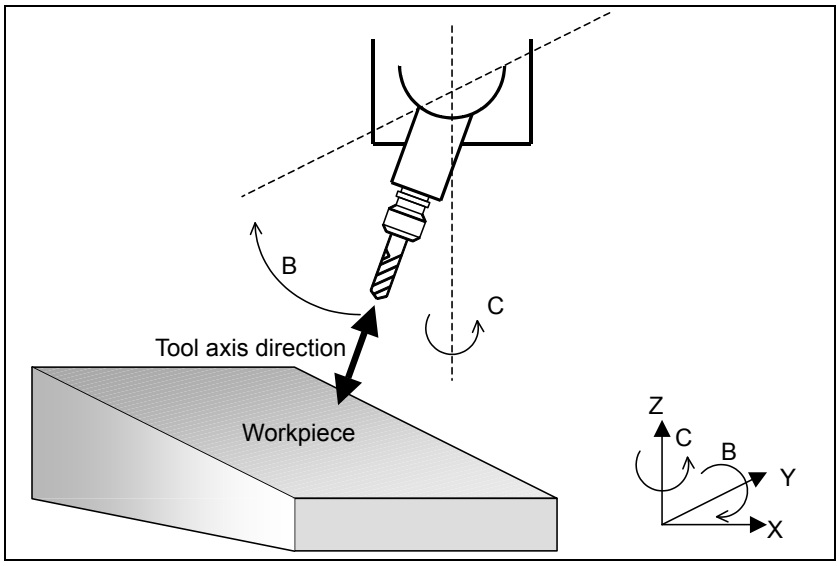
Overview

In the tool axis direction handle feed, tool axis direction JOG feed, and tool axis direction incremental feed, the tool or table is moved in the tool axis direction.

Explanation

- Tool axis direction

The tool axis direction that is taken when all the rotation axes for controlling the tool are at an angle of 0 degree is specified in parameters Nos.19697, 19698, and 19699. As the rotation axes for controlling the tool rotate, the tool axis direction changes according to the rotation axis angle.



- Tool axis direction feed in the tilted working plane command mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the tool axis direction feed in the tilted working plane command mode is assumed to be the Z direction in the feature coordinate system of the tilted working plane command.

- Tool axis direction handle feed

The tool axis direction handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.
<2> The tool axis direction feed mode signal (ALNGH) is set to "1" and the table base signal (TB_BASE) is set to "0".
<3> The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the tool axis direction handle feed mode effective is set in parameter No.12310.
<4> The value of parameter No.12310 matches the first manual handle feed axis selection signals (HS1A - HS1E).

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the tool axis direction by the amount of rotation.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.

- Tool axis direction JOG feed/tool axis direction incremental feed

The tool axis direction JOG feed or tool axis direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.
<2> The tool axis direction feed mode signal (ALNGH) is set to "1" and the table base signal (TB_BASE) is set to "0".
<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the axis corresponding to the direction specified by parameter No.19697. (Even when the tool axis direction is slant because of the settings of parameters Nos.19698 and 19699, the signal that activates the tool axis direction JOG feed or tool axis direction incremental feed is determined by parameter No.19697 only.)

Ex.) No.19697 = 3 (+Z-axis direction); Z-axis is the 3rd axis.

- +J3 : Tool axis direction +
- -J3 : Tool axis direction -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate of a rotation axis is the jog feedrate of the axis to be rotated (parameter No. 1423). The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

3.9.2 Tool Axis Right-Angle Direction Handle Feed / Tool Axis Right-Angle Direction JOG Feed / Tool Axis Right-Angle Direction Incremental Feed

Overview

In the tool axis right-angle direction handle feed, tool axis direction JOG feed, or tool axis direction incremental feed, the tool or table is moved in the tool axis direction.
If bit 1 (FLL) of parameter No. 12320 is set to 1, the tool or table is moved in the latitude or longitude direction determined by the tool axis direction vector.

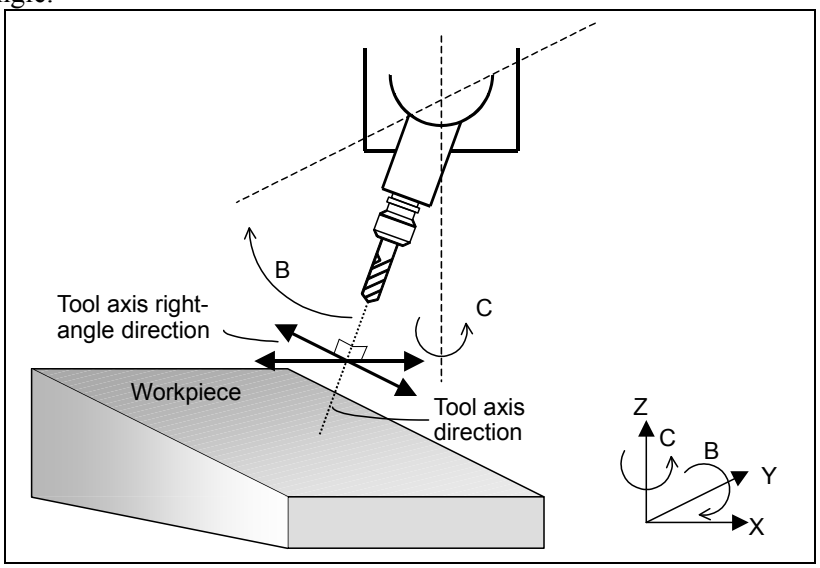
Explanation

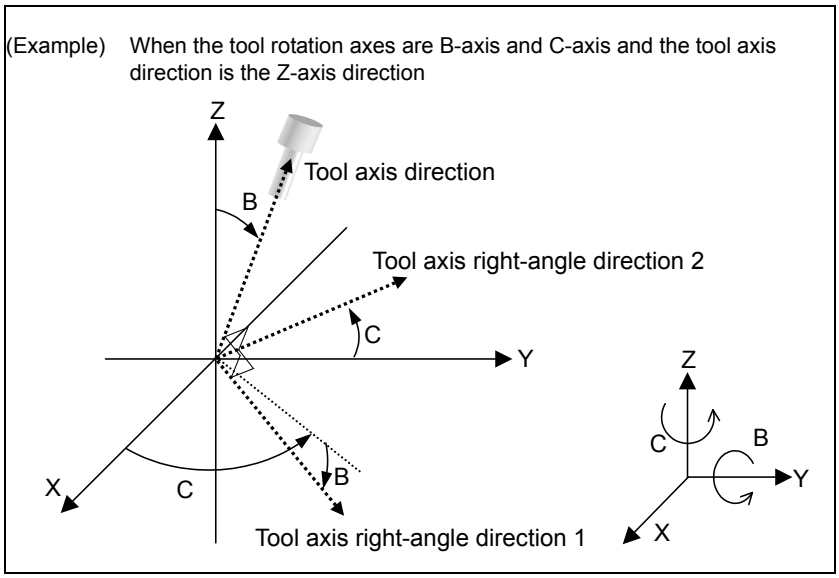
- Tool axis right-angle direction

There are two tool axis right-angle directions, which are perpendicular to the tool axis direction (see the previous section).

Parameter No.19697	Tool axis right-angle direction 1	Tool axis right-angle direction 2
1 (The reference tool direction is +X.)	+Y direction	+Z direction
2 (The reference tool direction is +Y.)	+Z direction	+X direction
3 (The reference tool direction is +Z.)	+X direction	+Y direction

This table shows the tool axis right-angle directions that may be taken when the angles of all the rotation axes for controlling the tool are 0 degree and when parameters Nos.19698 and 19699 are both set to 0.
When the reference tool axis direction is inclined based on the settings of parameters Nos.19698 and 19699, the tool axis right-angle direction is also inclined as much.
As the rotation axes for controlling the tool rotate, the tool axis right-angle direction changes according to the rotation axis angle.





- Latitude and longitude directions

When bit 1 (FLL) of parameter No. 12320 is set to 1, the feed direction is defined as follows:

Let a vector perpendicular to a plane formed by the tool axis direction vector ($\vec{T}$) and normal axis direction vector ($\vec{P}$) (parameter No. 12321) be the tool axis right-angle direction 1 (longitude direction) vector ($\vec{R}1$). When tool axis right-angle direction 1 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Longitude direction feed)

Equation: $\vec{R}1 = \vec{P} \times \vec{T}$

Let a vector perpendicular to the tool axis direction vector ($\vec{T}$) and tool axis right-angle direction 1 (longitude direction) vector ($\vec{R}1$) be the tool axis right-angle direction 2 (latitude direction) vector ($\vec{R}2$). When tool axis right-angle direction 2 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Latitude direction)

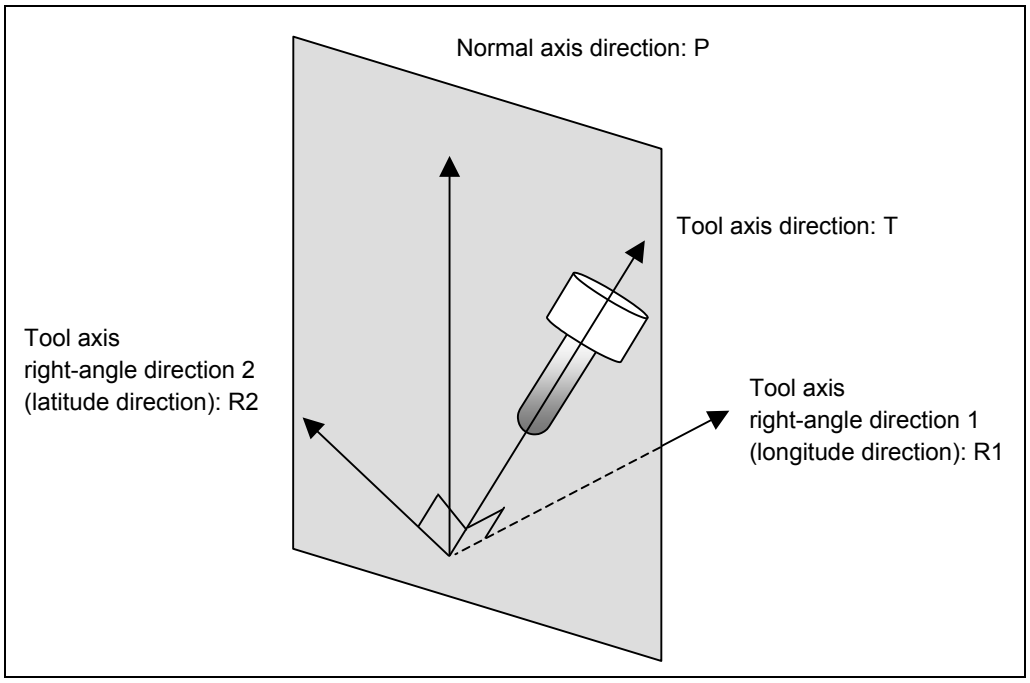
Equation: $\vec{R}2 = \vec{T} \times \vec{R}1$

When the tool axis direction vector ($\vec{T}$) is parallel to the normal axis direction vector ($\vec{P}$) (parameter No. 12321) (when the angle between them is not greater than the setting of parameter No. 12322), tool axis right-angle direction 1 and tool axis right-angle direction 2 are assumed as follows:

Parameter No.12321	Normal axis direction	Tool axis right-angle direction 1	Tool axis right-angle direction 2
1	+X direction	+Y direction	+Z direction
2	+Y direction	+Z direction	+X direction
3	+Z direction	+X direction	+Y direction

If 0 is set in parameter No. 12321, the normal axis direction is set to the reference tool axis direction (parameter No. 19697).

If a value other than 0 to 3 is specified in parameter No. 12321, alarm PS5459, “MACHINE PARAMETER INCORRECT” is issued.



- Tool axis right-angle direction feed in the tilted working plane command mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the tool axis right-angle direction feed in the tilted working plane command mode is defined as follows:

Tool axis right-angle direction 1:

X direction in the feature coordinate system of the tilted working plane command

Tool axis right-angle direction 2:

Y direction in the feature coordinate system of the tilted working plane command

- Tool axis right-angle direction handle feed

The tool axis right-angle direction handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.

<2> The tool axis right-angle direction feed mode signal (RGHTH) is set to "1" and the table base signal (TB_BASE) is set to "0".

<3> The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the tool axis right-angle direction handle feed mode effective is set in parameter No.12311 or No.12312.

<4> The value of parameter No.12311 or No.12312 matches the first manual handle feed axis selection signals (HS1A - HS1E).

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the tool axis right-angle direction by the amount of rotation.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.

- Tool axis right-angle direction JOG feed/tool axis right-angle direction incremental feed

The tool axis right-angle direction JOG feed or tool axis right-angle direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.

<2> The tool axis right-angle direction feed mode signal (RGHTH) is set to "1" and the table base signal (TB_BASE) is set to "0".
<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the axis corresponding to the direction that is perpendicular to the direction specified by parameter No.19697. (Even when the tool axis direction is slant because of the settings of parameters Nos.19698 and 19699, the signal that activates the tool axis right-angle direction JOG feed or tool axis right-angle direction incremental feed is determined by parameter No.19697 only.)
Ex.) No.19697=3 (+Z-axis direction); X-, Y-, and Z-axes are the 1st, 2nd, and 3rd axes respectively.

- +J1 : Tool axis right-angle direction 1 +
- -J1 : Tool axis right-angle direction 1 -
- +J2 : Tool axis right-angle direction 2 +
- -J2 : Tool axis right-angle direction 2 -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) for a driven feed axis direction selection signal. The manual feedrate override feature is available.

Feedrate clamp

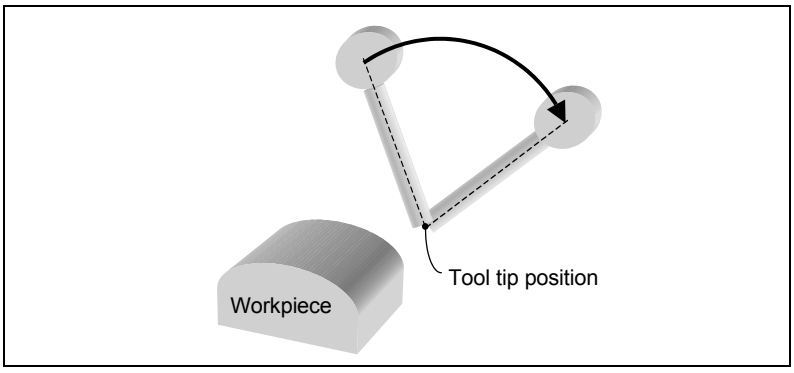
The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

3.9.3 Tool Tip Center Rotation Handle Feed / Tool Tip Center Rotation JOG Feed / Tool Tip Center Rotation Incremental Feed

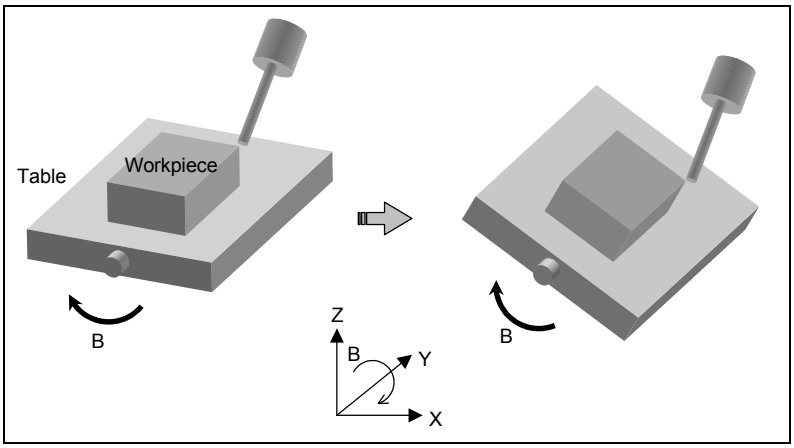
Overview

In the tool tip center rotation handle feed, tool tip center rotation JOG feed, and tool tip center rotation incremental feed, when a rotary axis is rotated by manual feed, the linear axes (X, Y, and Z axes) are moved so that turning the rotation axis does not change the relative relationship between the tool tip position and the workpiece (table).

- The following figure shows an example where the tool is rotated on the rotation axis. In this case, the linear axes are moved so that the position of the tool tip is not moved with respect to the workpiece.



- The following figure shows an example where the table is rotated on the rotation axis. As in the previous case, the linear axes are moved so that the position of the tool tip is not moved with respect to the workpiece (table).



- Tool tip center rotation handle feed

The tool tip center rotation handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.
<2> The tool tip center rotation feed mode signal (RNDH) is set to "1".
<3> The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the tool tip center rotation handle feed mode effective is set in parameter No.12313 or No.12314.
<4> The value of parameter No.12313 or No.12314 matches the first manual handle feed axis selection signals (HS1A - HS1E).

Amount of movement

When the manual pulse generator is rotated, the rotation axis is moved by the amount of rotation. The linear axes (X, Y, and Z axes) are moved so that turning the rotation axis does not change the relative relationship between the tool tip position and the workpiece.

Feedrate clamp

The feedrate is clamped so that the synthetic speed of the linear axes (in the tangential direction) does not exceed the manual rapid traverse rate (parameter No.1424) (of any moving linear axis). The feedrate is also clamped so that the speed of the rotation axis does not exceed the manual rapid traverse rate (parameter No.1424) (of that particular axis). Handle pulses generated while the clamp feedrate is exceeded are ignored.

- Tool tip center rotation JOG feed/tool tip center rotation incremental feed

The tool tip center rotation JOG feed or tool tip center rotation incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.
<2> The tool tip center rotation feed mode signal (RNDH) is set to "1".
<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the rotation axis to be rotated.
Ex.) When the B-axis (4th axis) is rotated
- +J4 : Tool tip center rotation feed +
- -J4 : Tool tip center rotation feed -

Feedrate

Control is exerted so that the synthetic speed of the linear axes (in the tangential direction) is the dry run rate (parameter No.1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate of a rotation axis is the jog feedrate of the axis to be rotated (parameter No. 1423). The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the synthetic speed of the linear axes (in the tangential direction) does not exceed the manual rapid traverse rate (parameter No.1424) (of any moving linear axis). The feedrate is also clamped so that the speed of the rotation axis does not exceed the manual rapid traverse rate (parameter No.1424) (of that particular axis).

- Selection of the tool length offset value

The tool length in 3-dimensional manual feed is determined as explained below. (Table 3.9.3 (a))
If bit 2 (LOD) of parameter No. 19746 is set to 0, the value set in parameter No. 12318 is assumed to be the tool length.
If the LOD parameter is set to 1, and the tool length offset function is performed, the offset data specified for the tool length offset is assumed to be the tool length.
If the LOD parameter is set to 1, and the tool length offset function is not performed, the tool length is determined as follows. If bit 3 (LOZ) of parameter No. 19746 is set to 0, the value set in parameter No. 12318 is assumed to be the tool length in 3-dimensional manual feed; if LOZ is set to 1, the tool length is assumed to be 0.

Table 3.9.3 (a) Tool length offset value in 3-dimensional manual feed

		Bit 2 (LOD) of parameter No. 19746		
		= 0	= 1	
			Tool length offset enabled	Tool length offset canceled
Bit 3 (LOZ) of parameter No. 19746	= 0	Parameter No. 12318	Offset data	Parameter No. 12318
	= 1			0

The tool length offset function is enabled when the following two conditions are both satisfied:

- The tool length offset function listed below is enabled (modal code of group 8 except G49)
  - G43/G44 : Tool length compensation
  - G43.4/G43.5 : Tool center point control
- The H/D code is other than 0.

If bit 6 (CLR) of parameter No. 3402 is set to 0 not to clear the tool length offset vector, G codes of group 8, and H codes at the time of a reset, the tool length offset status is maintained when a reset is made in the tool length offset mode.

3.9.4 Table Vertical Direction Handle Feed / Table Vertical Direction JOG Feed / Table Vertical Direction Incremental Feed

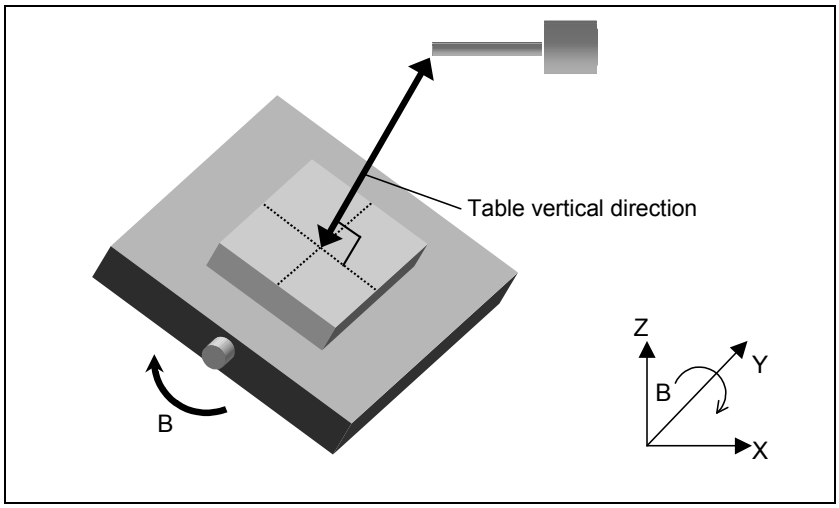
Overview

In the table vertical direction handle feed, table vertical direction JOG feed, and table vertical direction incremental feed, the tool is moved in the table vertical direction.

Explanation

- Table vertical direction

The table vertical direction is a direction vertical to the table. It is equal to the tool axis direction specified in parameter No.19697 when all of the rotation axes for controlling the table are at a an angle of 0 degree. When the rotation axes for controlling the table rotate, the table vertical direction changes according to the rotation axis angle.



- Table-based vertical direction feed in the tilted working plane command mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the table-based vertical direction feed in the tilted working plane command mode is assumed to be the Z direction in the feature coordinate system of the tilted working plane command.

- Table vertical direction handle feed

The table vertical direction handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.
<2> Both the tool axis direction feed mode signal (ALNGH) and the table base signal (TB_BASE) are set to "1".
<3> The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the table vertical handle feed mode effective is set in parameter No.12310.
<4> The value of parameter No.12310 matches the first manual handle feed axis selection signals (HS1A - HS1E).

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the table vertical direction by the amount of rotation.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.

- Table vertical direction JOG feed/table vertical direction incremental feed

The table vertical direction JOG feed or table vertical direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.
<2> Both the tool axis direction feed mode signal (ALNGH) and the table base signal (TB_BASE) are set to "1".
<3> The feed axis direction selection signal [+Jn,-Jn (where n = 1 to the number of controlled axes)] is set to "1" for the axis corresponding to the direction specified by parameter No.19697.
Ex.) No.19697 = 3 (+Z-axis direction); Z-axis is the 3rd axis.
- +J3 : Table vertical direction +
- -J3 : Table vertical direction -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.

If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) for a driven feed axis direction selection signal. The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

3.9.5 Table Horizontal Direction Handle Feed / Table Horizontal Direction JOG Feed / Table Horizontal Direction Incremental Feed

Overview

In the table horizontal direction handle feed, table horizontal direction JOG feed, and table horizontal direction incremental feed, the tool is moved in the table horizontal direction.
If bit 1 (FLL) of parameter No. 12320 is set to 1, the tool or table is moved in the latitude or longitude direction determined by the table-based vertical direction vector.

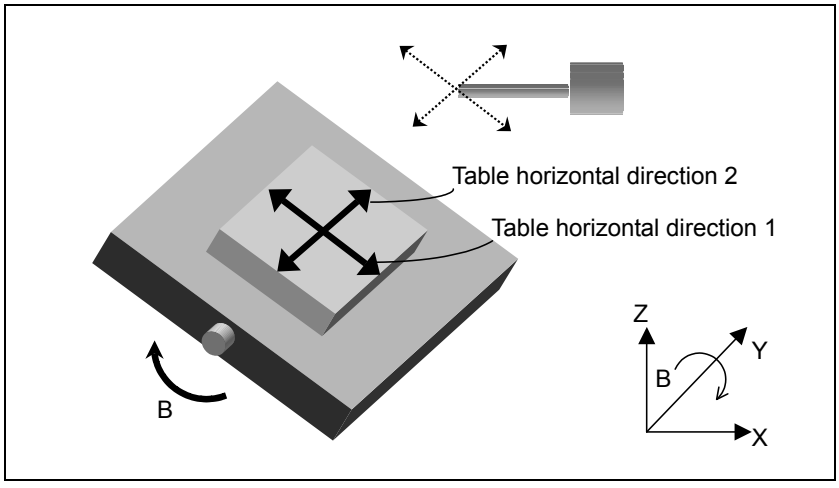
Explanation

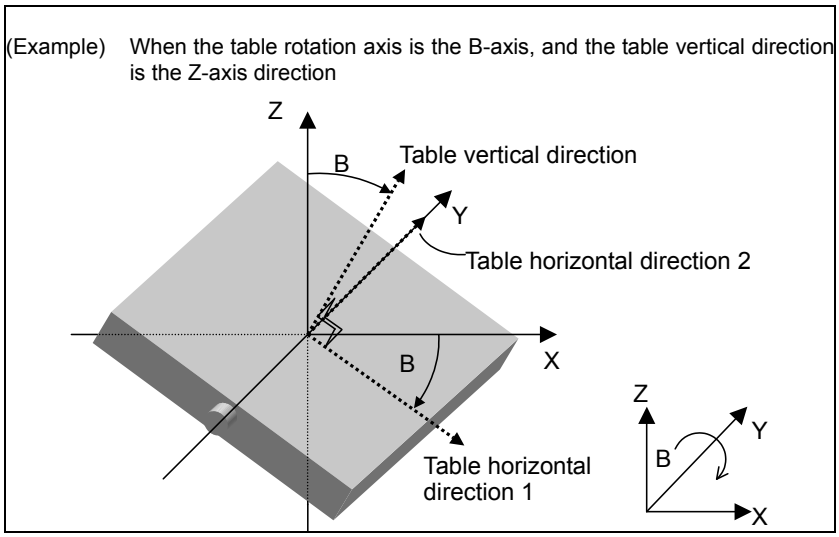
- Table horizontal direction

There are two table horizontal directions, which are perpendicular to the table vertical direction (see the previous section).

Parameter No.19697	Table horizontal direction 1	Table horizontal direction 2
1 (The reference tool direction is +X.)	+Y direction	+Z direction
2 (The reference tool direction is +Y.)	+Z direction	+X direction
3 (The reference tool direction is +Z.)	+X direction	+Y direction

This table shows the table horizontal directions that may be taken when the angles of all the rotation axes for controlling the table are 0 degree.
As the rotation axes for controlling the table rotate, the table horizontal direction changes according to the rotation axis angle.





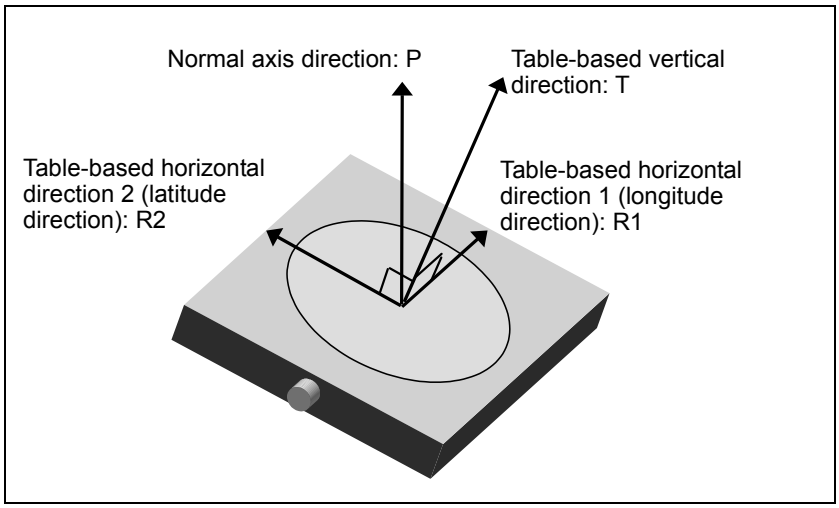
- Latitude and longitude directions

When bit 1 (FLL) of parameter No. 12320 is set to 1, the feed direction is defined as follows:
Let a vector perpendicular to a plane formed by the table-based vertical direction vector ($\vec{T}$) and normal axis direction vector ($\vec{P}$) (parameter No. 12321) be the table-based horizontal direction 1 (longitude direction) vector ($\vec{R1}$). When tool axis right-angle direction 1 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Longitude direction feed)
Equation: $\vec{R1} = \vec{P} \times \vec{T}$
Let a vector perpendicular to the table-based vertical direction vector ($\vec{T}$) and table-based horizontal direction 1 (longitude direction) vector ($\vec{R1}$) be the table-based horizontal direction 2 (latitude direction) vector ($\vec{R2}$). When tool axis right-angle direction 2 is selected, a movement in the positive direction means a movement in this vector direction, and a movement in the negative direction means a movement in the direction opposite to the vector direction. (Latitude direction)
Equation: $\vec{R2} = \vec{T} \times \vec{R1}$

When table-based vertical direction vector ($\vec{T}$) is parallel to the normal axis direction vector ($\vec{P}$) (parameter No. 12321) (when the angle between them is not greater than the setting of parameter No. 12322), tool axis right-angle direction 1 and tool axis right-angle direction 2 are assumed as follows:

Parameter No. 12321	Normal axis direction	Table-based horizontal direction 1	Table-based horizontal direction 2
1	+X direction	+Y direction	+Z direction
2	+Y direction	+Z direction	+X direction
3	+Z direction	+X direction	+Y direction

If 0 is set in parameter No. 12321, the normal axis direction is set to the tool axis direction.
If a value other than 0 to 3 is specified in parameter No. 12321, alarm PS5459 is issued.



- Table-based horizontal direction feed in the tilted working plane command mode

If bit 0 (TWD) of parameter No. 12320 is set to 1, the feed direction of the table-based horizontal direction feed in the tilted working plane command mode is defined as follows:

Table-based horizontal direction 1:

X direction in the feature coordinate system of the tilted working plane command

Table-based horizontal direction 2:

Y direction in the feature coordinate system of the tilted working plane command

- Table horizontal direction handle feed

The table horizontal direction handle feed is enabled when the following four conditions are satisfied:

<1> Handle mode is selected.

<2> Both the tool axis right-angle direction feed mode signal (RGHTH) and the table base signal (TB_BASE) are set to 1.

<3> The state of the first manual handle feed axis selection signals (HS1A - HS1E) to make the table horizontal direction handle feed mode effective is set in parameter No.12311 or No.12312.

<4> The value of parameter No.12311 or No.12312 matches the first manual handle feed axis selection signals (HS1A - HS1E).

Amount of movement

When the manual pulse generator is rotated, the tool is moved in the table horizontal direction by the amount of rotation.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424). Handle pulses generated while the clamp feedrate is exceeded are ignored.

- Table horizontal direction JOG feed/table horizontal direction incremental feed

The table horizontal direction JOG feed or table horizontal direction incremental feed is enabled when the following three conditions are satisfied:

<1> JOG mode or incremental feed mode is selected.

<2> Both the tool axis right-angle direction feed mode signal (RGHTH) and the table base signal (TB_BASE) are set to "1".

<3> The feed axis direction selection signal (+Jn, -Jn (where n = 1 to the number of controlled axes)) is set to "1" for the axis corresponding to the direction that is perpendicular to the direction specified by parameter No.19697.

Ex.) No.19697 = 3 (+Z-axis direction); X-, Y-, and Z-axes are the 1st, 2nd, and 3rd axes respectively.

- +J1 : Table horizontal direction 1 +
- -J1 : Table horizontal direction 1 -
- +J2 : Table horizontal direction 2 +
- -J2 : Table horizontal direction 2 -

Feedrate

The feedrate is the dry run rate (parameter No.1410). The manual feedrate override feature is available.
If bit 2 (JFR) of parameter No. 12320 is set to 1, the feedrate is the jog feedrate (parameter No. 1423) of a driven feed axis direction selection signal. The manual feedrate override feature is available.

Feedrate clamp

The feedrate is clamped so that the speed of each moving axis dose not exceed the manual rapid traverse rate (parameter No.1424).

Note

1 To perform a 3-dimensional handle feed, the manual handle feed option is required. To perform 3-dimensional handle interrupt, the manual handle interrupt option is required.
2 When a 3-dimensional handle interrupt is performed, rotation axis command execution must not be in progress in automatic operation.
3 When the manual reference position return mode is selected, 3-dimensional manual feed is not enabled.
4 If per-axis interlock is enabled to at least one of 3-dimensional manual feed axes, movement with manual feed is not performed.
5 When the offset value specified for the tool length offset function is used for tool center point rotation feed (when bit 2 (LOD) of parameter No. 19746 is set to 1), the controlled point should generally be shifted. (Set bit 5 (SVC) of parameter No. 19665 to 1.)
In this case, specify the tool length with a radius value.

3.10 DISTANCE CODED LINEAR SCALE INTERFACE

Overview

The interval of each reference marks of distance coded linear scale are variable. Accordingly, if the interval is determined, the absolute position can be determined. The CNC measures the interval of reference marks by axis moving of short distance and determines the absolute position. Consequently the reference position can be established without moving to reference position.

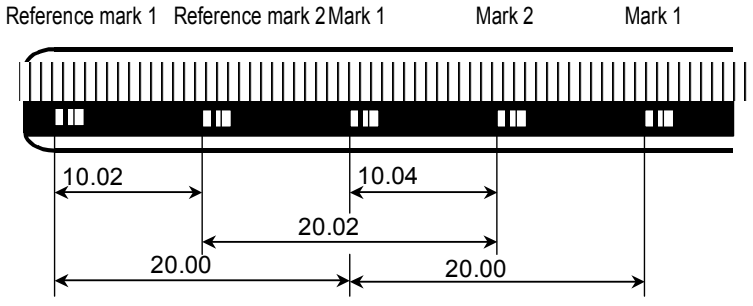


Fig. 3.10 (a) Example of distance coded linear scale

This is an optional function.

3.10.1 Procedure for Reference Position Establishment

Procedure

(1) Select the JOG mode, and set the manual reference position return selection signal ZRN to "1".
(2) Set a direction selection signal(+J1,-J1,+J2,-J2,…) for a target axis.
(3) The axis is fed at a constant low speed (reference position return FL feedrate specified by parameter No.1425 setting).
(4) When a reference mark is detected, the axis stops, then the axis is fed at a constant low speed again.
(5) Above (4) is executed repeatedly until two, three or four reference marks are detected. And absolute position is determined and reference position establishment signal (ZRF1,ZRF2,ZRF3, …) turns to "1". (A number of reference marks is determined by bit 2 (DC2x) and 1 (DC4x) of parameter No.1802.)

The timing chart for this procedures is given below.

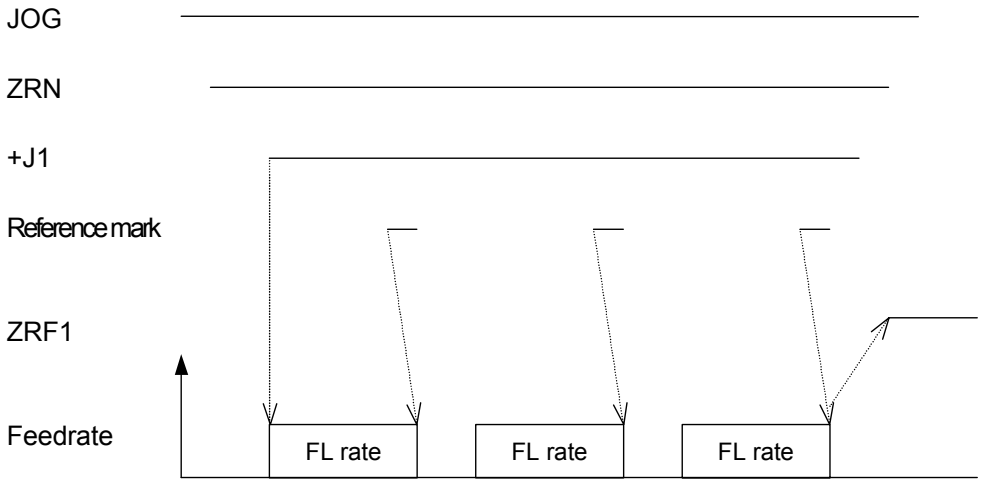


Fig. 3.10.1 (a) Timing chart for reference position establishment

- Procedure for establishing a reference position through automatic operation

If an automatic reference position return (G28) is specified before a reference position is not established, steps (3) to (5) above are performed automatically.
After the reference position is established, the automatic reference position return is performed.

- Stopping the operation for establishing a reference position

The operation for establishing a reference position is stopped if any of
the following operations is performed in steps (3) to (5), described above.

- Reset
- Setting the feed axis direction selection signal (+J1, -J1, +J2, -J2, etc.) to 0

If any of the following operations is performed during the operation of automatic reference position return (G28) before a reference position is not established, the operation for establishing a reference position stops:

- Reset
- Performing feed hold during movement from an intermediate position

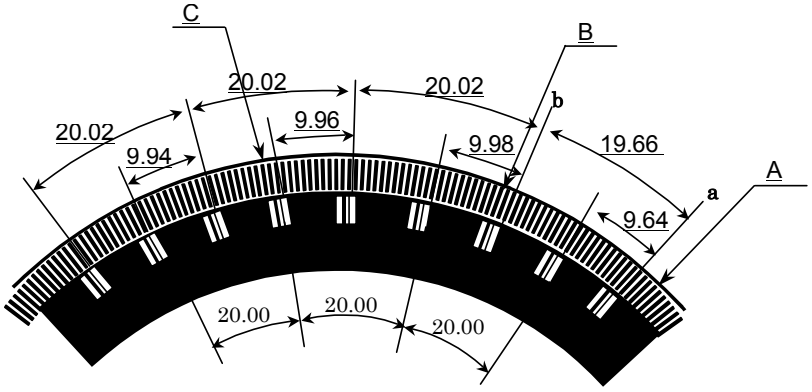
If the operation for establishing a reference position is stopped by an operation other than a reset, the operation for establishing a reference position must be reset and resumed.

3.10.2 Reference Position Return

(1) When the reference position is not established and the axis moved by turning the feed axis direction signal (+J1,-J1,+J2,-J2,...) to "1" in REF mode, the reference position establishment procedure is executed.
(2) When the reference position is already established and the axis is moved by turning the feed axis direction signal (+J1,-J1,+J2,-J2,...) to "1" in REF mode, the axis is moved to the reference point without executing the reference position establishment procedure.
(3) When the reference position is not established and the reference position return command (G28) is executed, the reference position establishment procedure is executed. The next movement the axis depends on the setting of bit 0 (RFS) of parameter No. 1818.
(4) When the reference position is already established and the reference position command (G28) is executed, the movement of the axis depends on the setting of bit 1 (RF2) of parameter No.1818.

3.10.3 Distance Coded Rotary Encoder

In case of setting a rotary axis, if bit 3 (DCRx) of parameter No.1815 is set, the setting axis is regarded as being equipped with a distance coded rotary encoder.
In case of distance coded rotary encoder, the marker interval may be different from parameter setting value. (a-b section of the following figure) When the reference point return is executed through this section, it is not able to establish the reference point. Therefore, in case of distance coded rotary encoder, if the reference point return is started for B point from A point of below figure, the reference point is not established yet at B point. The reference point return is re-started for C point. The reference point return procedure is finished at C point.



- When the reference point return procedure is executed, the coordinate value are rounded in 0 to 360 degree, even if Machine coordinate values are linear axis type.
- In case of distance coded rotary encoder, only the measurement by three points or four points is possible. (bit 2 (DC2) of parameter No.1802 is disregarded as 0.)

3.10.4 Axis Synchronization Control

Requirements when this function is used with axis synchronization control axes

When this function is used with axis synchronization control axes, the distance coded linear scale used for the master axis and that used for the slave axis must have reference marks placed at identical intervals.
(Set identical values in parameters Nos. 1821 and 1882 for both the master and slave axes.)
This function does not work unless the use of this function is specified for both the master and slave axes (bit 2 (DCL) of parameter No. 1815 is 1).
Also, in all parameters related to this function, except parameter No. 1883, 1884 (distance from the scale zero point to reference position 1, 2), set identical values for both the master and slave axes.

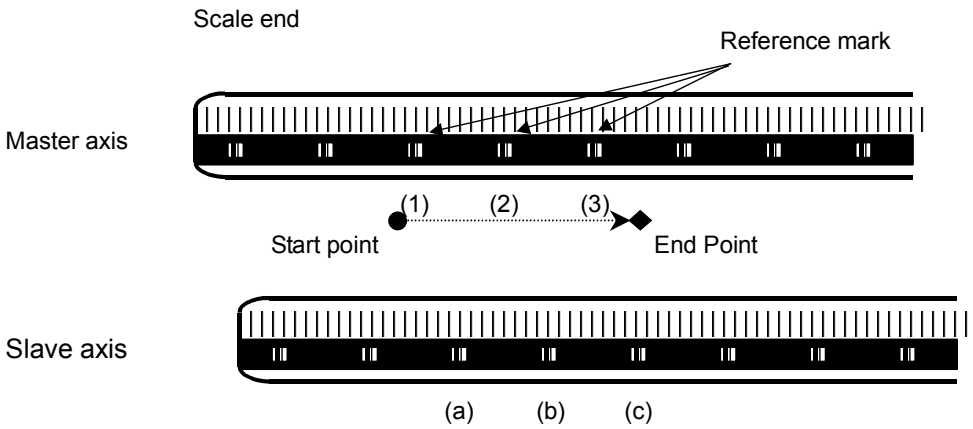
If a parameter value for the master axis differs from the corresponding parameter value for the slave axis, alarm SV1051, “ILLEGAL SYNCHRONOUS AXIS” is issued.

NOTE
When this function is used with axis synchronization control axes for which the operation mode is switched between synchronization operation and normal operation, this function is enabled only if the synchronization select signal (SYNC1, SYNC2, ...) is 1. (During establishment of a reference position, the synchronization select signal status must be maintained.)

Reference position establishment with axis synchronization control axes

With axis synchronization control axes, a reference position is established as follows. When a reference mark for the master or slave axis is detected, a stop takes place temporarily. Then, a feed operation is performed again at the reference position return FL feedrate. This sequence is repeated until a reference mark is detected three or four times for both the master and slave axes. Then the absolute position is calculated for both the master and slave axes, and the reference position establish signals ZRF1, ZRF2, ... are set to 1.
After the reference position has been established by the above operation, a synchronization error is corrected. (Checking for excessive synchronization error alarm 2 is made even during reference position establishment.)

(Example of 3 points measurement system)



In the above example, the following sequence is executed.

a. When the reference mark (1) of the master axis is detected, both master axis and slave axis stop.
b. Both the axes begin to move again at a reference position return FL feedrate.
c. When the reference mark (a) of the slave axis is detected, both axes stop again.
d. Both the axes begin to move again at FL feedrate.
e. Both axes repeat the operation until all point ((2) -> (b) ->(3) -> (c)) are detected.
f. When the slave axis detects the third reference mark (c), both the axes end the reference position establishment.

NOTE
In case of this function is used with axis synchronization control axes, if the value of parameters Nos. 1883 and 1884 for both the master and slave axes is 0, the reference position is not established. Also, the reference position establish signals ZRF1, ZRF2, ... are set to 0.

3.10.5 Axis Control by PMC

In PMC axis control, if the reference position return command (axis control command code 05H) is issued for an axis having a distance coded linear scale, reference position return is performed according to the reference position return sequence for the distance coded linear scale.
Specifically, the following operations take place:

Before reference position establishment	The reference position is established by detecting two, three or four reference marks. Movement to the reference position is not performed.
After reference position establishment	Positioning at the reference position is performed.

3.10.6 Angular Axis Control

There are the following limitations when the angular axis control is used.

(a) It is necessary to use the linear scale with the distance coded reference mark for both the perpendicular axis and the angular axis.

(b) When the reference point of the perpendicular axis is established, it is necessary to establish the reference point of the angular axis previously. When the reference point of the angular axis is not previously established, the alarm DS0020, “REFERENCE RETURN INCOMPLETE” is generated.

(c) During the reference point establishment operation of the angular axis, the command in the perpendicular axis is invalid in the manual reference point return.

3.10.7 Note

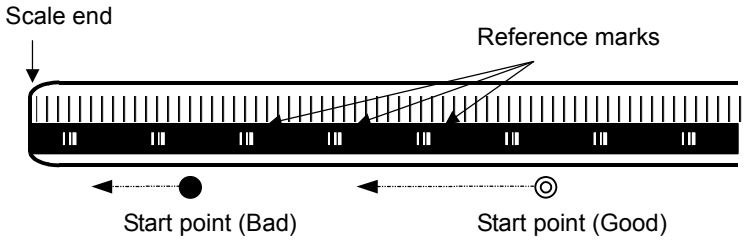
(1) In the case of the actual interval of reference marks is different from parameter setting value, the alarm DS1449, “REFERENCE MARKS ARE DIFFERENT FROM PARAMETER” occurs.

(2) This function is disabled if any of the following conditions is satisfied:
- Either parameter No.1821 (mark-1 interval) or parameter No.1882 (mark-2 interval) is set to 0.
- The setting of parameter No.1821 is greater than or equal to the setting of parameter No.1882.
- The difference between the settings made for parameters Nos. 1821 and 1882 is greater than or equal to twice either setting.
- The absolute-position detection function is enabled. (Bit 5 (APCx) of parameter No.1815 is set to 1.)

(3) A difference of parameters Nos.1821 and 1882 must be more than 4.
Example)
When the scale, which is that mark1 interval is 20.000mm and mark2 interval is 20.004mm, is used on IS-B machine :
When the detection unit of 0.001mm is selected, parameter No.1821 and No.1882 must be set "20000" and "20004", and the difference of them is "4".
To use such a scale, please adjust the detection unit by modification of parameters No.1820 (CMR) and Nos.2084/2085(flexible feed gear) to make the difference of parameters Nos.1821 and 1882 more than 4 as following examples.
(a) Set the detection unit=0.0001mm, and set No.1821=200000, No.1882=200040
(b) Set the detection unit=0.0005mm, and set No.1821=40000, No.1882=40008

NOTE
When the detection unit is changed, parameters relating to the detection unit (such as the effective area and positional deviation limit) must also be changed accordingly.

(4) In this procedure, the axis does not stop until two, three or four reference marks are detected. If this procedure is started at the position near the scale end, CNC can not detect three or four reference marks and the axis does not stop until over travel alarm occurs. Please care to start at the position that has enough distance from scale end.



(5) When the axis used this function, the following function can not be used.
- Absolute position detection (absolute pulse coder)
- 3-dimensional error correction

(6) If axial movement is made in the direction opposite to that of reference position return, the movement is reversed to the direction of reference position return after three or four reference marks have been detected. Steps (3) to (5) of the basic procedure for establishing a reference position are carried out to establish the reference position.

(7) Straightness compensation function
When the reference point establishment of moving axis is executed after the establishment of compensation axis, the compensation axis is moved by straightness compensation amount when the reference point of moving axis is established.

(8) The reference point establishment is not performed during synchronous control is activated.

(9) The reference point establishment is not performed during composite control is activated.

(10) The reference point establishment is not performed during superimposed control is activated.

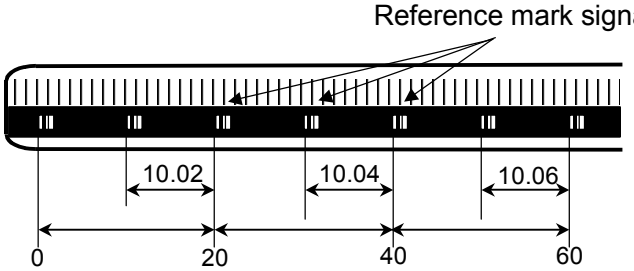
3.11 LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL)

Overview

By using High-resolution serial output circuit for the linear scale with distance-coded reference marks (serial), the CNC measures the interval of referenced mark by axis moving of short distance and determines the absolute position.
This function enables high-speed high-precision detection by using High-resolution serial output circuit.
It is available that using maximum stroke 30 meters length.

Explanation

The linear scale with distance-coded reference marks (serial) is combined the irregular reference marked linear scale with the High-resolution serial output circuit, it can detect the accurate position.

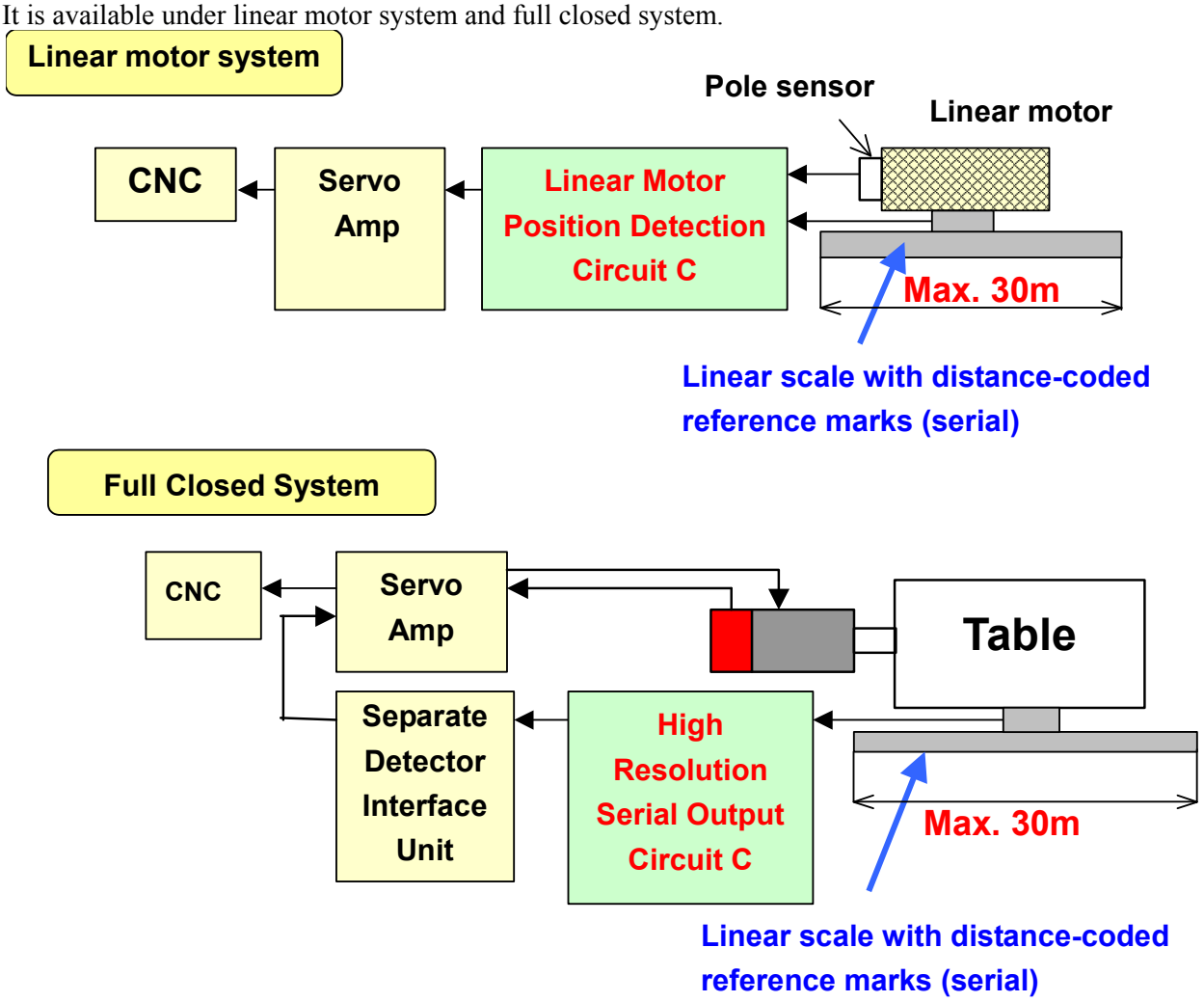


The CNC measures the interval of referenced mark by axis moving of short distance and determines the absolute position, because of the interval of each reference mark is different with regular interval.
It is not necessary that the axis is moved to the reference position for establishment of reference position.

This function enables high-speed high-precision detection by using High-resolution serial output circuit.
It is available that using maximum stroke 30 meters length.

- Connection

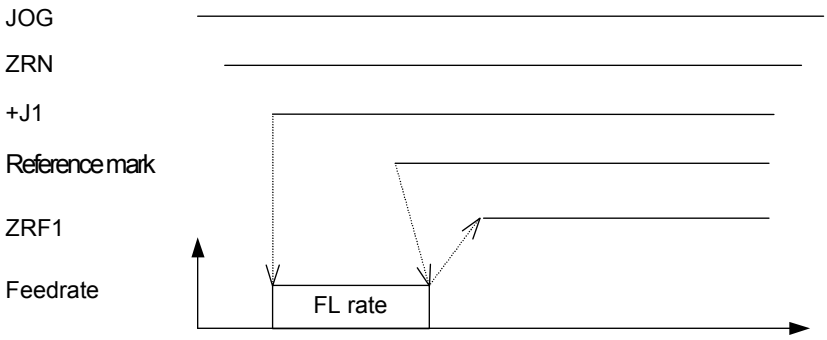
It is available under linear motor system and full closed system.



- Procedure for reference position establishment through manual operation

(1) Select the JOG mode, and set the manual reference position return selection signal ZRN to "1".
(2) Set a direction selection signal(+J1,-J1,+J2,-J2,…) for a target axis.
(3) The axis is fed at a constant low speed (reference position return FL feedrate specified by parameter No.1425 setting).
(4) When the absolute position of linear scale with distance-coded reference marks (serial) is detected, the axis stops. Then the absolute position of CNC is calculated and reference position establishment signal (ZRF1,ZRF2,ZRF3,…) turns to "1".

The timing chart for this procedures is given below.



- Procedure for reference position establishment through automatic operation

If an automatic reference position return (G28) is specified before a reference position is not established, steps (3) to (4) above are performed automatically.
After the reference position is established, the automatic reference position return is performed by setting of bit 0 (RFS) of parameter No. 1818.

- Stopping the operation for establishing a reference position

The operation for establishing a reference position is stopped if any of
the following operations is performed in steps (3) to (4), described above.

- Reset
- Setting the feed axis direction selection signal (+J1, -J1, +J2, -J2, etc.) to 0
- Setting the Servo off signals (SVF1, SVF2, etc.) to 1

If any of the following operations is performed during the operation of automatic reference position return (G28) before a reference position is not established, the operation for establishing a reference position stops:

- Reset
- Performing feed hold during movement from an intermediate position
- Setting the Servo off signals (SVF1, SVF2, etc.) to 1

If the operation for establishing a reference position is stopped by an operation other than a reset, the operation for establishing a reference position must be reset and resumed.

- Establishing a reference position and moving to the reference position

By following operation, establishing a reference position and moving to the reference position is performed.

	Moving through manual operation in REF mode	Moving through automatic operation by automatic reference position return (G28)
The reference position is not established.	Establishing the reference position	Firstly, moving to the intermediate position, and establishing the reference position. Secondly, whether moving to the reference position or not is performed by setting bit 0 (RFS) of parameter No.1818.
The reference position is established.	Moving to the reference position	Whether moving to the intermediate position and the reference position or not is performed by setting bit 1 (RF2) of parameter No.1818.

- Feed axis synchronization control

In case of using the axis synchronization control, please confirm the following items.

- When this function is used with axis synchronization control axes, the linear scale with distance-coded reference marks (serial) used for the master axis and that used for the slave axis must have reference marks placed at identical intervals.
- The master axis scale and the slave axis scale should be installed in parallel direction. (The zero positions should be faced the same direction.)

- To the parameters, which relate to this function (except No.1883, No.1884), the same value must be set for the master axis and for the slave axis.
- The linear scale with distance-coded reference marks (serial) should be applied for the master axis and the slave axis.
  If either of the master axis or the slave axis is not the linear scale with distance-coded reference marks (serial), alarm DS0018, "SERIAL DCL:MISMATCH(SSYNC CTRL)" occurs when reference position establishment is tried.
- During operating the establishment of reference position, the state of signal (SYNCn or SYNCJn) for selecting synchronized axis should be kept.

Procedure for reference position establishment by axis synchronization control is as follows.

- Both of axes (master axis and slave axis) are fed on the reference position return FL feedrate until distance coded scales of both axes detect the absolute position.
- Then absolute position of both axes are calculated and Reference Position Establishment Signals (ZRF1,ZRF2,...) turn to "1".

- Angular axis control

In case of using the angular axis control, please confirm the following items.

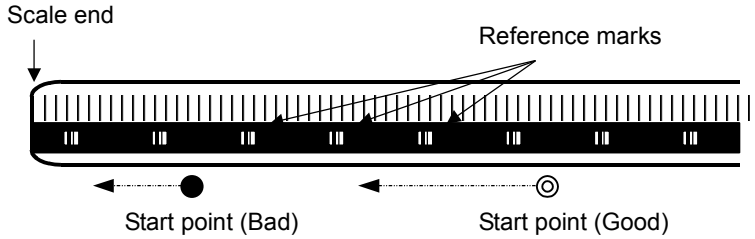
- It is necessary to use the linear scale with distance-coded reference marks (serial) for both the perpendicular axis and the angular axis.
  If not, the alarm DS0019, "SERIAL DCL:MISMATCH(ANGL-AXIS)" occurs when reference position establishment is commanded.
- When the reference point establishment of angular and perpendicular axes are tried, please set bit 2 (AZR) of parameter No. 8200 to '0' and input signal NOZAGC to '0'.
  If not, the alarm DS0019 occurs when reference position establishment is commanded.
- When the reference point of the perpendicular axis is established, it is necessary to establish the reference point of the angular axis previously. When the reference point of the angular axis is not previously established, the alarm DS0020 occurs.
- On angular axis control, if you use automatic setting of parameter No.1883,1884 on reference point establishment (bit 2 (DATx) of parameter No.1819 = 1), please establish reference point of perpendicular axis after reference point establishment and return of angular axis.

In manual reference position return, the perpendicular axis cannot be specified while the angular axis reference point is being established. The perpendicular axis, if specified, is ignored.

⚠ CAUTION
1 When the Linear scale with distance-coded reference marks (serial) is used, please set bit 3 (SDCx) of parameter No.1818 to 1.

⚠ CAUTION

2 On the Linear scale with distance-coded reference marks (serial), the axis does not stop until three reference marks are detected. If this procedure is started at the position near the scale end, CNC can not detect three reference marks and the axis does not stop until over travel alarm occurs. Please care to start at the position that has enough distance from scale end.
And if establishment of reference position is failed, the establishment is retried. Then axis does not stop until still more three reference marks are detected. So please set the maximum move amount (detection unit : parameter No.14010) not to reach the scale end.



3 On flexible synchronization control mode, reference position can't be established.

4 Straightness compensation function
When the reference point establishment of moving axis is executed after the establishment of compensation axis, the compensation axis is moved by straightness compensation amount when the reference point of moving axis is established.

5 It is not available to use this function and the temporary absolute coordinate setting together.

6 The reference point cannot be established during twin table synchronous operation or single operation.

7 Angular axis control cannot be performed together with synchronous control, composite control, or superposition control.

4 AUTOMATIC OPERATION

Programmed operation of a CNC machine tool is referred to as automatic operation.
This chapter explains the following types of automatic operation:

4.1 MEMORY OPERATION 1007
Operation by executing a program registered in CNC memory
4.2 MDI OPERATION 1009
Operation by executing a program entered from the MDI unit
4.3 DNC OPERATION.....1012
Function for executing a program while reading the program from an input device connected to the RS232C interface or a memory card.
4.4 SCHEDULE OPERATION 1015
Function for executing a program while reading the program from an input device connected to the RS232C interface or a memory card according to a schedule.
4.5 EEXTERNAL SUBPROGRAM CALL (M198) 1018
Function for calling and executing subprograms (files) registered in an external input/output device during memory operation
4.6 EXTERNAL SUBPROGRAM CALLS USING THE DATA SERVER AVAILABLE IN MULTI-PATH SYSTEMS 1022
Function for enabling external subprogram call commands using the Data Server to be specified simultaneously from multiple paths in a multi-path system
4.7 MANUAL HANDLE INTERRUPTION.....1023
Function for performing manual feed during movement executed by automatic operation
4.8 MIRROR IMAGE.....1030
Function for enabling mirror-image movement along an axis during automatic operation
4.9 PROGRAM RESTART 1031
Function for restarting machining from a block by specifying the sequence number or block number of that block
4.10 QUICK PROGRAM RESTART.....1051
Function for restarting machining from a block by specifying that block on the relevant screen
4.11 TOOL RETRACT AND RECOVER.....1080
Function for retracting the tool to a desired position
4.12 MANUAL INTERVENTION AND RETURN 1089
Function for returning to the pre-intervention position in the event of manual intervention during a feed hold stop and resuming automatic operation
4.13 RETRACE 1092
Function for executing a program in the reverse direction.
4.14 ACTIVE BLOCK CANCEL FUNCTION.....1101
Function for stopping operation and canceling the remaining movement by inputting a signal during automatic operation.

4.1 MEMORY OPERATION

Programs are registered in memory in advance. When one of these programs is selected and the cycle start switch on the machine operator's panel is pressed, automatic operation starts, and the cycle start LED goes on.
When the feed hold switch on the machine operator's panel is pressed during automatic operation, automatic operation is stopped temporarily. When the cycle start switch is pressed again, automatic operation is restarted.

When the 

key on the MDI unit is pressed, automatic operation terminates and the reset state is entered.

For the multi-path control, the programs for the multiple paths can be executed simultaneously so the multiple paths can operate independently at the same time.
The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

Memory operation

Procedure

1 Press the MEMORY mode selection switch.
2 Select a program from the registered programs. To do this, follow the steps below.
  2-1 Press  key to display the program screen.
  2-2 Press address  key.
  2-3 Enter a program number using the numeric keys.
  2-4 Press the [O SRH] soft key.
3 For the multi-path control, select the path to be operated with the path selection switch on the machine operator's panel.
4 Press the cycle start switch on the machine operator's panel.
  Automatic operation starts, and the cycle start LED goes on. When automatic operation terminates, the cycle start LED goes off.
5 To stop or cancel memory operation midway through, follow the steps below.
  a. Stopping memory operation
    Press the feed hold switch on the machine operator's panel. The feed hold LED goes on and the cycle start LED goes off. The machine responds as follows:
    (i) When the machine was moving, feed operation decelerates and stops.
    (ii) When dwell was being performed, dwell is stopped.
    (iii) When M, S, or T was being executed, the operation is stopped after M, S, or T is finished.
    When the cycle start switch on the machine operator's panel is pressed while the feed hold LED is on, machine operation restarts.
  b. Terminating memory operation
    Press the  key on the MDI unit.
    Automatic operation is terminated and the reset state is entered.
    When a reset is applied during movement, movement decelerates then stops.

Explanation

- Memory operation

After memory operation is started, the following are executed:
(1) A one-block command is read from the specified program.
(2) The block command is decoded.
(3) The command execution is started.
(4) The command in the next block is read.
(5) Buffering is executed. That is, the command is decoded to allow immediate execution.
(6) Immediately after the preceding block is executed, execution of the next block can be started. This is because buffering has been executed.
(7) Hereafter, memory operation can be executed by repeating the steps (4) to (6).

- Stopping and terminating memory operation

Memory operation can be stopped using one of two methods: Specify a stop command, or press a key on the machine operator's panel.

- The stop commands include M00 (program stop), M01 (optional stop), and M02 and M30 (program end).
- There are two keys to stop memory operation: The feed hold key and reset key.

- Program stop (M00)

Memory operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged as in single block operation. The memory operation can be restarted by pressing the cycle start button. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

- Optional stop (M01)

Similarly to M00, memory operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator's panel is set to ON. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

- Program end (M02, M30)

When M02 or M30 (specified at the end of the main program) is read, memory operation is terminated and the reset state is entered.
Since it depends on machine specification whether the active line returns to top of program or not when M30 is read, refer to the manual provided by the machine tool builder for details.

- Feed hold

When Feed Hold button on the operator's panel is pressed during memory operation, the tool decelerates to a stop at a time.

- Reset

Automatic operation can be stopped and the system can be made to the reset state by using 

key on the MDI unit or external reset signal. When reset operation is applied to the system during a tool moving status, the motion is slowed down then stops.

- Optional block skip

A block skip slash (/) must be specified at the beginning of a block.
Note that if a slash is placed elsewhere, the information from the slash to immediately before the EOB code is ignored.

- Cycle start for the multi-path control

For the multi-path control, a cycle start switch is provided for each path. This allows the operator to activate a single paths to operate them at the same time in memory operation or MDI operation. In general, select the path to be operated with the path selection switch on the machine operator's panel and then press the cycle start button to activate the selected path. (The procedure may vary with the machine tool builder. Refer to the appropriate manual issued by the machine tool builder.)

4.2 MDI OPERATION

In the MDI mode, a program consisting of up to 511 characters can be created in the same format as normal programs and executed from the MDI unit.
MDI operation is used for simple test operations.
The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

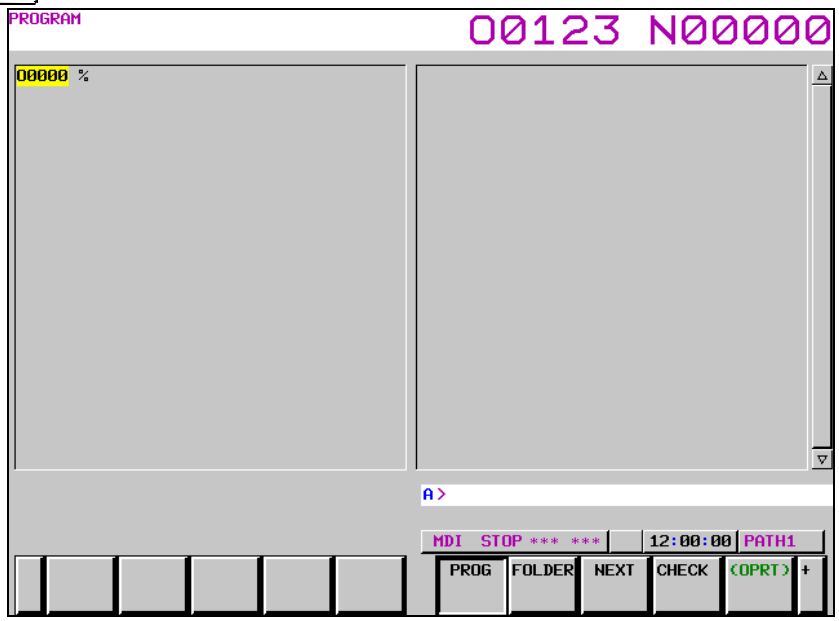
MDI Operation

Procedure

1 Select the MDI mode.

For the multi-path control, select the path for which a program is to be created with the path selection switch. Create a separate program for each path.

2 Press the PROG key to select the program screen. The following screen appears:



MDI program screen

At this time, program number "O0000" is inserted automatically.

3 Prepare a program to be executed by an operation similar to normal program editing. M99 specified in the last block can return control to the beginning of the program after operation ends. Word insertion, modification, deletion, word search, address search, and program search are available for programs created in the MDI mode.

4 To entirely erase a program created in MDI mode, use one of the following methods:

a. Enter address P 0 key, then press the DELETE key.

b. Alternatively, press the RESET key. In this case, set bit 7 (MCL) of parameter No. 3203 to 1 in advance.

5 To execute a program, set the cursor on the head of the program.
Push Cycle Start button on the operator's panel. By this action, the prepared program will start.
(For the multi-path control, select the path to be operated with the path selection switch on the machine operator's panel beforehand.) When the program end (M02, M30) or ER(%) is executed, the prepared program will be automatically erased and the operation will end.
By command of M99, control returns to the head of the prepared program.

6 To stop or terminate MDI operation in midway through, follow the steps below.

a. Stopping MDI operation
Press the feed hold switch on the machine operator's panel. The feed hold LED goes on and the cycle start LED goes off. The machine responds as follows:
(i) When the machine was moving, feed operation decelerates and stops.
(ii) When dwell was being performed, dwell is stopped.
(iii) When M, S, or T was being executed, the operation is stopped after M, S, or T is finished.
When the cycle start switch on the machine operator's panel is pressed, machine operation restarts.

b. Terminating MDI operation
Press the RESET key.
Automatic operation is terminated and the reset state is entered.

When a reset is applied during movement, movement decelerates then stops.

Explanation

The previous explanation of how to execute and stop memory operation also applies to MDI operation, except that in MDI operation, M30 does not return control to the beginning of the program (M99 performs this function).

- Erasing the program

Programs prepared in the MDI mode will be erased in the following cases:

- In MDI operation, if M02, M30 or ER(%) is executed.
- When bit 6 (MER) of parameter No. 3203 is set to 1, and the last block of the program is executed in single block operation

NOTE
In the two cases above, program erasure can be prevented by setting bit 6 (MKP) of parameter No. 3204 to 1.

- In MEM mode, if memory operation is performed.
- In EDIT mode, if any editing is performed.
- When the  and  keys are pressed.
- Upon reset when bit 7 (MCL) of parameter No.3203 is set to 1

NOTE
Upon reset when the parameter MCL = 0, the cursor moves to the end of the program.

- Restart

If a program is not executed even once after the program is input, the program is executed from the beginning, regardless of where the cursor is placed. However, a program is executed starting at the beginning of the block where the cursor is place, if the program is stopped for a reason such as single block operation after restart of an MDI operation then is restarted after an editing operation.

⚠ CAUTION
When an MDI program is restarted, the program is executed starting at the beginning of the block where the cursor is placed, regardless of the cursor position in the block.
(Example)

When the cursor is placed on G90

```
        :
G91 X100.0 G90Y200.0 Z300.0 ;
        :
```

The program is executed starting at the beginning (namely, G91) of this block. So, the tool moves by 100.0 along the X-axis in the incremental programming, and moves to 200.0 and 300.0 along the Y-axis and Z-axis, respectively, in the absolute programming.

- Editing a program during MDI operation

A program can be edited during MDI operation. By setting bit 5 (MIE) of parameter No. 3203 to 1, editing can be disabled. However, even when bit 5 (MIE) of parameter No. 3203 is set to 1, editing can be enabled by resetting the operation.

- Absolute/incremental command

When bit 4 (MAB) of parameter No. 3401 is set to 1, the absolute/incremental programming of MDI operation does not depend on G90/G91. In this case, the incremental programming is set when bit 5 (ABS) of parameter No. 3401 is set to 0, and the absolute programming is set when bit 5 (ABS) of parameter No. 3401 is set to 1.

Bit 4 (MAB) of parameter No. 3401=0	Bit 4 (MAB) of parameter No. 3401=1	
	Bit 5 (ABS) of parameter No. 3401=0	Bit 5 (ABS) of parameter No. 3401=1
Absolute mode operation with G90 command, and incremental programming operation with G91 command	Incremental mode operation at all times, independent of G90/G91 command	Absolute mode operation at all times, independent of G90/G91 command

NOTE
When G code system A is used on a lathe system, the parameters MAB and ABS are invalid.

Limitation

- Program registration

Programs created in MDI mode cannot be registered.

- Number of characters in a program

A created program can consist of up to 511 characters including "O0000" automatically inserted.

- Subprogram nesting

The subprogram call command (M98) can be described in a program created in MDI mode. That is, programs that are registered in memory through MDI operation can be called and executed. The level of subprogram call nesting is the same as in MEM operation.

- Macro call

When the custom macro function is enabled, a macro program can be created and executed even in the MDI mode. Moreover, a macro program can be called for execution.

NOTE
The GOTO statement, WHILE statement, and DO statement cannot be executed in a program created in the MDI mode. An alarm PS0377, “ILLEGAL MODE FOR GOTO/WHILE/DO” is issued.
When a program including those statements is to be executed, register the program in the program memory then call the program for execution.

4.3 DNC OPERATION

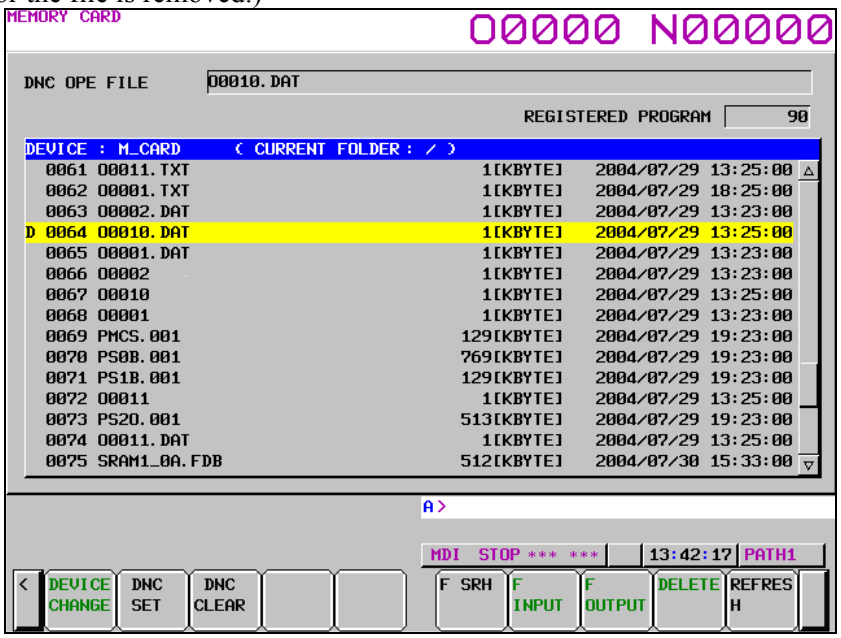
By activating automatic operation during the DNC operation mode (RMT), it is possible to perform machining (DNC operation) while a program is being read in via RS232C interface. Available devices include a memory card and Data Server.
To use the DNC operation function, it is necessary to set the parameters related to the RS232C interface in advance.
The procedure described below is just an example. For actual operation, refer to the relevant manual of the machine tool builder. For a Data Server, also refer to the “Fast Ethernet/Fast Data Server OPERATOR’S MANUAL”.

NOTE
DNC operation cannot be performed using a program in a USB memory.

DNC operation

Procedure

1 Press the REMOTE switch on the machine operator's panel to enter the DNC mode.
2 Select the program to be executed.
- Selecting a DNC operation file
  On the memory card (or floppy cassette) list screen, move the cursor to the file to be subjected to DNC operation and press "DNC SET" to select the file to be subjected to DNC operation. (The selected file is marked with "D".)
- Releasing a DNC operation file
  On the memory card (or floppy cassette) list screen, move the cursor to the file for which DNC operation is released and press "DNC CLEAR" to release the DNC operation file. (The "D" mark for the file is removed.)



3 Press the cycle start switch to execute the selected file. For details on the REMOTE switch, refer to the manual provided by the machine tool builder.

4 During DNC operation, executed programs are listed on the program check screen and program screen.

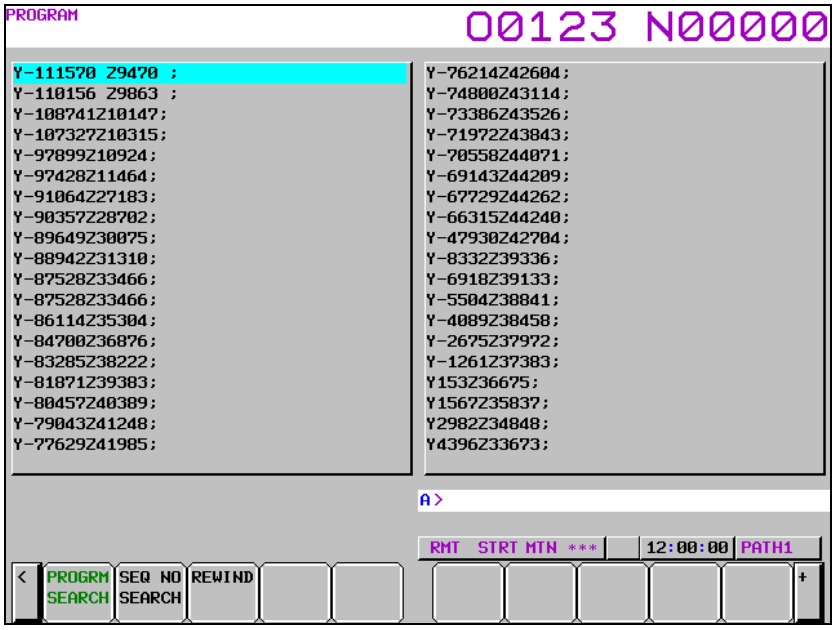


Fig. 4.3 (a) PROGRAM screen

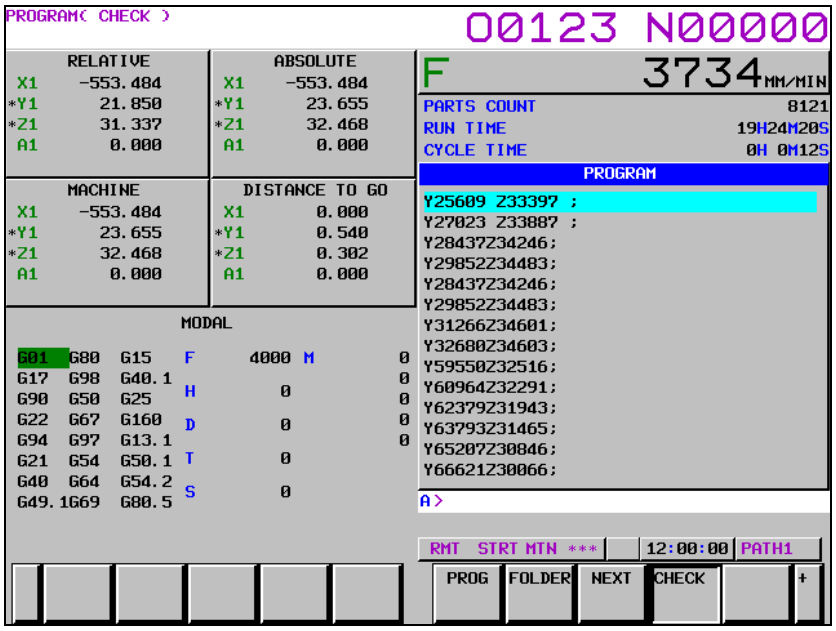


Fig. 4.3 (b) PROGRAM CHECK screen

NOTE
1 Before selecting a DNC operation file, be sure to release all schedule data. DNC operation and schedule operation cannot be specified at the same time.
2 A DNC operation file cannot be released during DNC operation.
3 To switch between devices when DNC settings are made, release the settings and then make the settings again.

Explanation

During DNC operation, subprograms and macro programs stored in memory can be called.

Limitation

- **M198 (command for calling a program from within an external input/output unit)**

In DNC operation, M198 cannot be executed. If M198 is executed, alarm PS0210, “CAN NOT COMMAND M198/M99” is issued.

- **Custom macro**

In DNC operation, custom macros can be specified, but no repeat instruction and branch instruction can be programmed. If a repeat instruction or branch instruction is executed, alarm PS0123, “ILLEGAL MODE FOR GOTO/WHILE/DO” is issued.

- **M99**

For returning from a subprogram or macro program to the calling program during DNC operation, the specification of a return command (M99P...) with a sequence number specified is not allowed.

4.4 SCHEDULE OPERATION

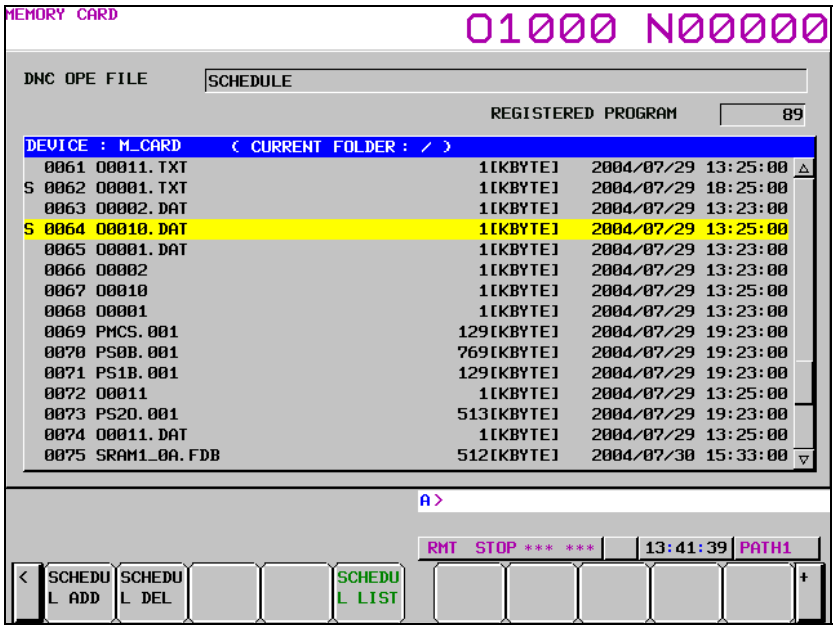
To perform schedule operation, select files (programs) registered in a memory card and specify the sequence of execution and the repetition count of each program.

NOTE
Schedule operation cannot be performed using a program in a USB memory.

Schedule operation

Procedure

1 Press the REMOTE switch on the machine operator's panel to enter the DNC mode.
2 Select the program to be subjected to schedule operation.
- Setting a schedule
  Move the cursor to the file to be subjected to schedule operation and press the soft key [SCHEDUL ADD]. The files registered as schedule data are marked with "S" on the left sides of their file names. The files are subjected to schedule operation in the order in which the files are selected. When settings are made this way, the repetition count of execution of each file is set to 1. To change the repetition count or execution order or check the schedule, press the soft key [SCHEDUL LIST] to display the schedule list screen.



[SCHEDUL ADD]

Adds the file at the cursor position as schedule data.

[SCHEDUL DEL]

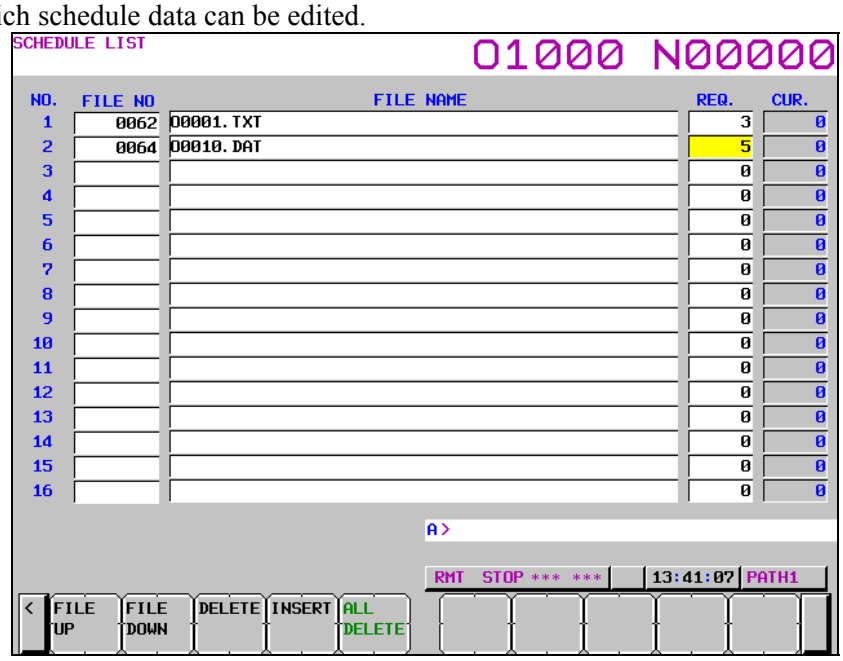
Deletes the file at the cursor position from schedule data when the file is registered as schedule data.

[SCHEDUL LIST]

Lists the settings of schedule data to edit the repetition count or the like (see the next page for details).

- Editing a schedule

  To edit schedule data, press the soft key [SCHEDUL LIST] to display the schedule list screen, on which schedule data can be edited.



[FILE UP] Moves the file at the cursor position up one line and moves the replaced file down one line.

[FILE DOWN] Moves the file at the cursor position down one line and moves the replaced file up one line.

[DELETE] Deletes the file at the cursor position and moves the files below the cursor up one line.
[INSERT] Moves the files below the cursor down one line.
[ALL DELETE] Deletes all records.

3 Press the cycle start switch to execute the selected files. For details on the REMOTE switch, refer to the manual provided by the machine tool builder.

NOTE
1 Before setting schedule operation, release DNC operation files in the MDI mode. DNC operation and schedule operation cannot be specified at the same time.
2 Before starting schedule operation, confirm that schedule data is set correctly on the schedule list screen.
3 Schedule data cannot be changed or edited during schedule operation. Before changing schedule data, make a reset to stop operation.

Restrictions

- Repetition count

The maximum repetition count during schedule operation is 9999. When a negative value is specified, an endless loop (LOOP display) is assumed. The file for which 0 is set is skipped and processing proceeds to the next file.

- Number of registered files

The maximum number of programs that can be registered as schedule setting data is 20.

- Selectable files

The files to be selected as schedule setting data must be registered in the same folder. (Files in the different folders cannot be selected.)

- M code

Even if a code other than M02 and M30 in the execution program is executed, the current count on the schedule execution status screen is not increased.

- Floppy disk folder display during execution of a file

During schedule operation, folders in a floppy disk cannot be displayed in a background edit.

- Intervention during automatic operation

Intervention in schedule operation cannot be performed during automatic operation.

- During multi-path control

The scheduling function cannot be used by multiple paths at the same time.

4.5 EXTERNAL SUBPROGRAM CALL (M198)

During memory operation, you can call and execute a subprogram registered in an external device (such as a Memory Card, Handy File, or Data Server) connected to the CNC.

NOTE
A program in a USB memory cannot be called using an external subprogram call (M198).

Format

M198 Pxxxxxxxx Lyyyyyyyy ;

↑ ↑

Pxxxxxxxx : Program number (or file number)
Lyyyyyyyy : Number of repetitive calls

- **If calling with a file name**

M198 <xxxx> Lyyyyyyyy ;

↑ ↑

xxxx : File name of an external subprogram
Lyyyyyyyy : Number of repetitive calls

When address L is omitted, the number of repetitive calls is assumed to be 1.

FS16 compatible command format
(The following command format is valid only when a 4-digit program number is used.)

M198 Pxxxx yyyy ;

↑ ↑

xxxx : Number of repetitive calls
yyyy : Program number (or file number)
When the number of repetitive calls is omitted, it is assumed to be 1.

Explanation

M code M198 specifies an external subprogram call. You can also call an external subprogram using an M code set in parameter No. 6030. (When an M code other than M198 is set as an M code for calling an external subprogram, M198 is executed as a normal M code.)

Specify a program number (file number) registered in an external device at address P. If the specified program number (file number) is not registered in the connected external device, an alarm PS1079, "PROGRAM FILE NOT FOUND" is issued.

If calling with a file name, the file name of an external subprogram must be specified immediately after M198.
If, in an external subprogram call specified with a file name, no program is registered in the connected external device, alarm SR1079 (alarm SR1966, "FILE NOT FOUND (MEMORY CARD)" in the case of a memory card) is issued.

Characters that can be used in the file name of an external subprogram and the number of characters in a file name are as described below.

- The following characters can be used in a file name:
  Alphabetical characters (uppercase and lowercase letters), numeric characters, and the four symbols below:
  - + _ .
  Because "." and ".." are reserved file names, they cannot be used.
- The maximum number of characters that can be used in a file name in each external device, as well as the alarms that are issued if an item out of the range (such as a character that cannot be specified) is specified, is as given below.

External device name	Number of characters	Alarm issued if an item out of the range is specified
Handy File	17	SR1079
FLOPPY CASSTTE	17	SR1079
Memory Card	12	SR1968
Data Server	32	PS0311

Example 1)

If using a Data Server as the external device, specify the command for calling the external subprogram located on the Data Server, as follows:

M198 <SMPL100**> L**10; Calls SAMPL100 ten times.

File name on the Data Server: SMPL100

```
%
<SMPL100>（Sample Program1）;
N1 G55;
N2 G90 G01 X0 Y0 Z30.0 F1000;
N3 G01 X0 Y0 Z0 F1000;
N4 ……;
```

Example 2)

If using a memory card as the external device, specify the command for calling the external subprogram located on the memory card, as follows:

M198 <SMPL200**> L**20; Calls SAMPL200 twenty times.

File name on the memory card: SMPL200

```
%
O0200（Sample Program2）;
N1 G55;
N2 G90 G01 X0 Y0 Z30.0 F1000;
N3 G01 X0 Y0 Z0 F1000;
N4 ……;
```

NOTE

1 If the file name of an external subprogram differs from its program name, as in the example below, specify the file name (SMPL200) on the Memory Card.

NOTE

2 If an MS-DOS format floppy disk is used with a Handy File, a file name can be specified as follows:

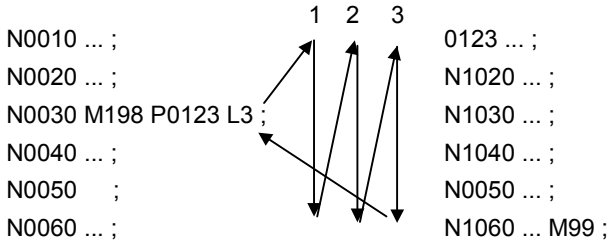
- A file name of up to 12 characters, 8 characters + "." + 3characters, can be specified. For an external subprogram call, there are cases in which no alarm is issued even if a call is made with a specification exceeding 12 characters, as in the following example.
  Example) M198 <ABCDEFGH>
  For this command, the file with the file name, "ABCDEFGH.DAT", is the one to be called.
  Even if <ABCDEFGH I JKLMN> is specified as a file name, the file with the first 8 characters + "DAT", i.e., "ABCDEFGH.DAT", is the one to be called.

Example 3)

M198 P0123 L3;

This command specifies that the subprogram having external subprogram number O0123 is to be called three times repeatedly.

The subprogram is called from the main program and executed as follows:



- **Program number call**
- **File name call**

You can also specify a subprogram call with its program number instead of the file number by the setting of bit 2 (SBP) of parameter No. 3404.

- **Program number O8-digit**

The called file name is as follows.

Table 4.5 (a) The file name called when instructing in M198

Device	Program number is 7-digit or less. (example: M198P1234567)	Program number is 8-digit. (example: M198P12345678)
When an external device is excluding a fast data server, or is a fast data server excluding in the storage mode.	“O” plus a 7-digit numeric (example: O1234567)	a 8-digit numeric (example: 12345678)
When an external device is a fast data server in the storage mode.	“O” plus a 8-digit numeric (example: O01234567)	a 8-digit numeric (example: 12345678)

NOTE

1 An external subprogram call can be specified during program operation in the MEM/MDI mode. However, in case of using with MDI mode, it is necessary to set the parameter MDE (No.11630#1)=1.

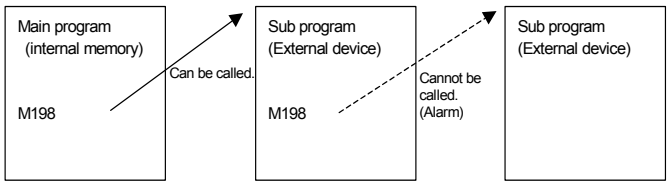
2 An external subprogram call is available for the following external devices:

External device name	Program number call	File number call	File name call
Handy File	Available	Available	Available
FLOPPY CASSTTE	Available	Available	Available
Memory Card	Available	Unavailable	Available
Data Server	Available	Unavailable	Available

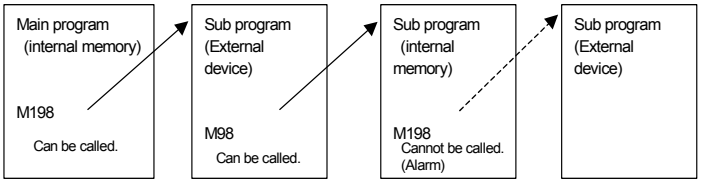
3 To perform a subprogram call using a Memory Card as the external device, set bit 7 (MNC) of parameter No. 138 to 1 and I/O channel (parameter No. 0020) to 4.
A program number call is always enabled regardless of the setting of bit 2 (SBP) of parameter No. 3404.

4 To perform a file name call using a Handy File or floppy cassette as the external device, set bit 2 (SBP) of parameter No. 3404 to 1.

5 called using another external device subprogram call. (An alarm PS1080, “DUPLICATE DEVICE SUB PROGRAM CALL” is issued.)



6 A subprogram registered in internal memory can be called from a subprogram called using an external device subprogram call. From the called subprogram in internal memory, another external device subprogram call cannot be performed. (An alarm (PS1080) is issued.)



7 A call using the external device subprogram call function is counted as one level of subprogram nesting.

8 In a multi-path system, an external device subprogram call cannot be performed simultaneously from multiple paths.
When external subprogram calls using the Data Server that are available in multi-path systems are used, however, external subprogram call commands using the Data Server can be specified simultaneously from multiple paths.

4.6 EXTERNAL SUBPROGRAM CALLS USING THE DATA SERVER AVAILABLE IN MULTI-PATH SYSTEMS

Overview

In a multi-path system, external subprogram call commands using the Data Server can be specified simultaneously from multiple paths.

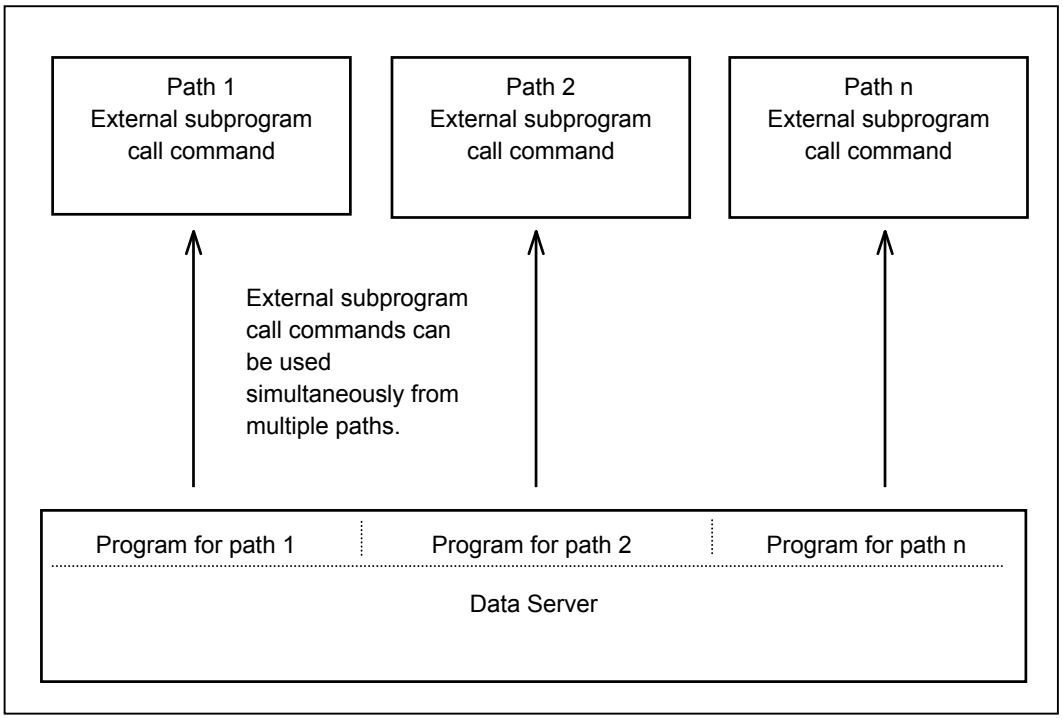


Fig. 4.6 (a) External subprogram calls specified simultaneously from multiple paths

Explanation

In a multi-path system, external subprogram call commands using the Data Server can be specified simultaneously from multiple paths.

・Example) Specifying an external subprogram call simultaneously from two paths

During the execution of an external subprogram command (M198) in path 1, another external subprogram call command (M198) can be executed from path 2, as shown below.

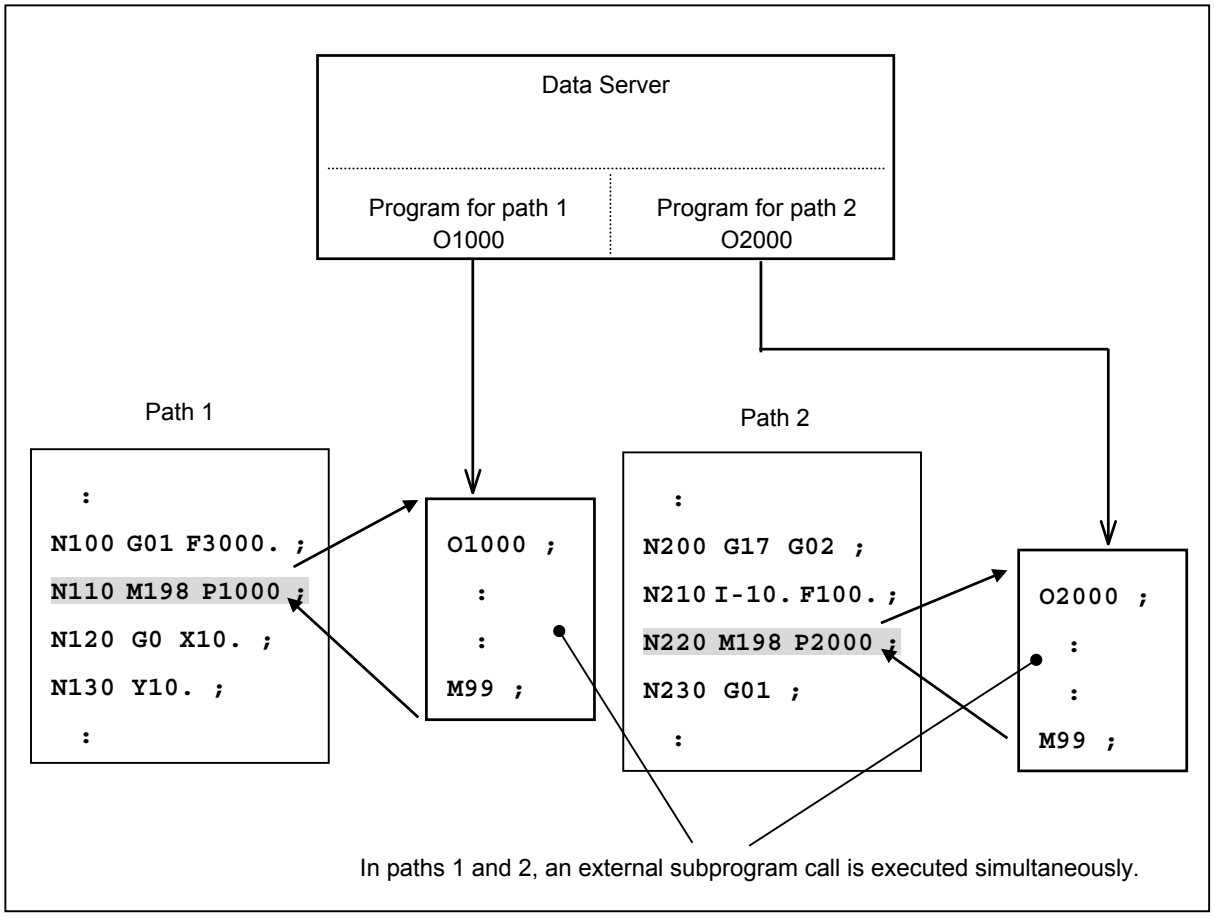


Fig. 4.6 (b) Specifying an external subprogram call in paths 1 and 2 simultaneously

Note

NOTE

1 This function is enabled only when the Data Server is selected as a foreground input device.
Other devices such as a memory card cannot be used to execute an external subprogram call command simultaneously in multiple paths.
2 DNC operation cannot be performed simultaneously in multiple paths.
3 The same external subprogram cannot be executed simultaneously in multiple paths.
4 Before specifying an external subprogram command, set a folder for performing an external subprogram call. For details, refer to the following manual:
”Fast Ethernet / Fast Data Server OPERATOR’S MANUAL (B-64014EN)”
5 To use this function, set bit 0 (ESB) of parameter No. 8107 to 1.

4.7 MANUAL HANDLE INTERRUPTION

By rotating the manual pulse generator in the automatic operation mode (manual data input, DNC operation, or memory operation) or in the memory editing mode, handle feed can be superimposed on movement by automatic operation. A handle interruption axis is selected using the manual handle interruption axis selection signals HsnIA to HSnIE.
The minimum unit of travel distance per scale division is the least input increment. One of four types of magnifiers selected with manual handle feed selection signals MP1 and MP2 can be applied. With bit 3 (HIT) of parameter No. 7103, the minimum unit of travel distance can be further 10 times greater.

A handle feed magnifier is selected using the manual handle feed amount selection signal. (See "MANUAL HANDLE FEED".)

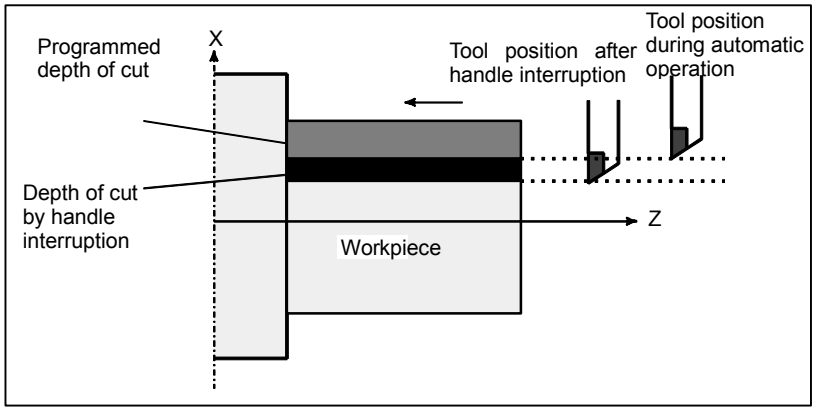


Fig. 4.7 (a) Manual handle interruption

> ⚠ **WARNING**
> The travel distance per scale division by manual handle interruption is the least input increment as with manual handle feed. With a machine of metric input and inch output, for example, the travel distance per 254 scale divisions is 0.01 inch. With a machine of inch input and metric output, the travel distance per 100 scale divisions is 0.254 mm.

Explanation

- Interruption operation

1 When the handle interruption axis selection signal for a handle interruption axis is set to 1 in the automatic operation mode (manual data input, DNC operation, or memory operation) or in the memory editing mode, manual handle interruption can be performed by rotating the handle of the manual pulse generator.

> **NOTE**
> Even when the feedrate override signal sets 0%, manual handle interruption can be accepted.

2 For the method of selecting a manual handle interruption axis, refer to the relevant manual of the machine tool builder.

3 The feedrate during manual handle interruption is the sum of feedrate used for automatic operation and the feedrate used for movement by manual handle interruption. However, the feedrate during manual handle interruption is controlled so that it does not exceed the maximum allowable cutting feedrate for the axis.

Example

Suppose that the maximum allowable cutting feedrate for an axis is 5 m/min, and that a movement is made in the + direction at 2 m/min along the axis. In this case, manual handle interruption can be accepted even when the manual pulse generator is rotated up to a speed equivalent to 3 m/min. Manual handle interruption by rotation in one direction can be accepted even when the manual pulse generator is rotated to a speed equivalent to 7 m/min.
If the manual pulse generator is rotated to a speed beyond the upper limits, those pulses from the manual pulse generator that correspond to the excess are lost, resulting in a mismatch between the scale mark of the manual pulse generator and the actually interrupted travel distance.

4 For a magnifier for manual handle interruption, refer to the relevant manual of the machine tool builder.

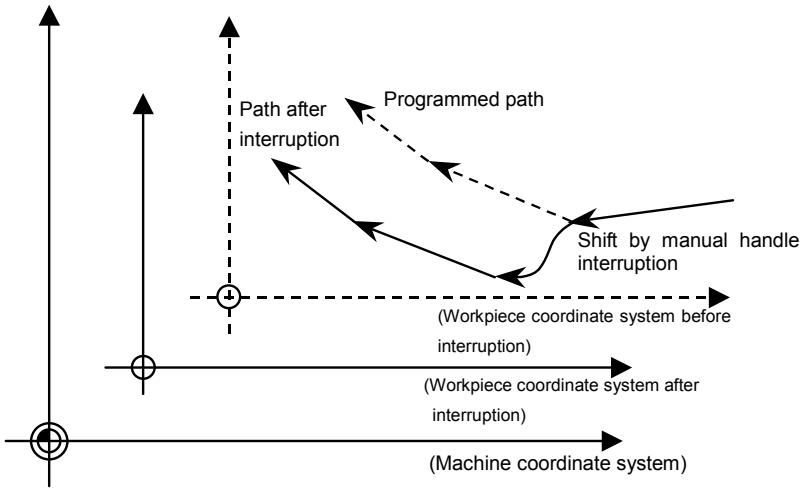
5 If the travel direction is reversed as a result of manual handle interruption, backlash compensation is performed. Pitch error compensation is performed for the position after interruption.

6 In manual handle interruption, only acceleration/deceleration for cutting feed is enabled. By setting bit 0 (MNJ) of parameter No. 1606 to 1, acceleration/deceleration for both of cutting feed and jog feed can be applied to manual handle interruption.

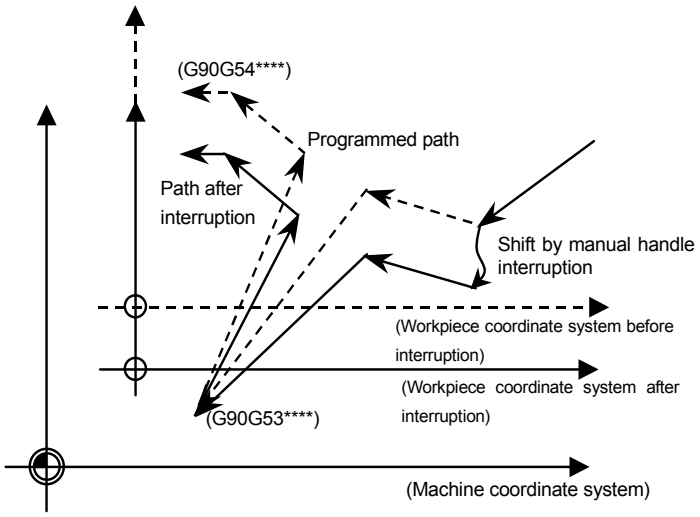
- Manual handle interruption and coordinate system

1 The amount of manual handle interruption shifts the workpiece coordinate systems and the local coordinate system. So, the machine moves, but the coordinates in the workpiece coordinate systems and the local coordinate system remain unchanged.
Regardless of which coordinate system is selected, all workpiece coordinate systems and the local coordinate system shift by the same amount.

- Absolute coordinates
  → Remain unchanged by handle interruption.
- Relative coordinates
  → Change by the amount of handle interruption.
- Machine coordinates
  → Change by the amount of handle interruption.



2 Even when manual handle interruption is performed, the machine coordinate system remains unchanged. The absolute command (G53) in the machine coordinate system is not affected by manual handle interruption.

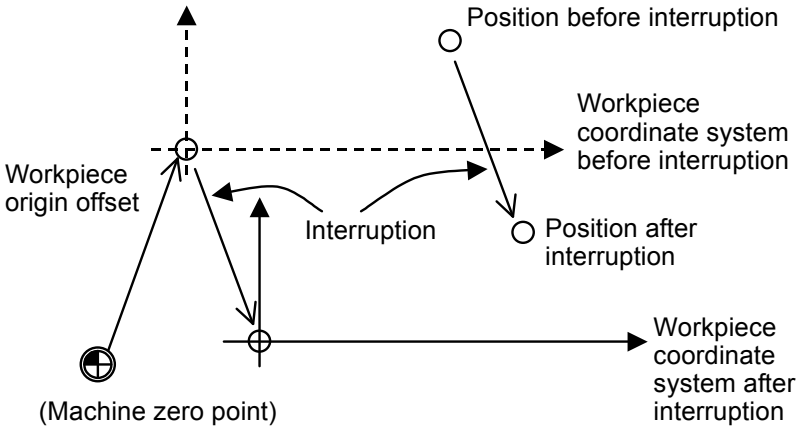


3 In automatic reference position return (G28), the end point (reference position) is not affected by manual handle interruption. However, the midpoint is in the workpiece coordinate system, so that the position shifted by the amount of interruption becomes the midpoint.

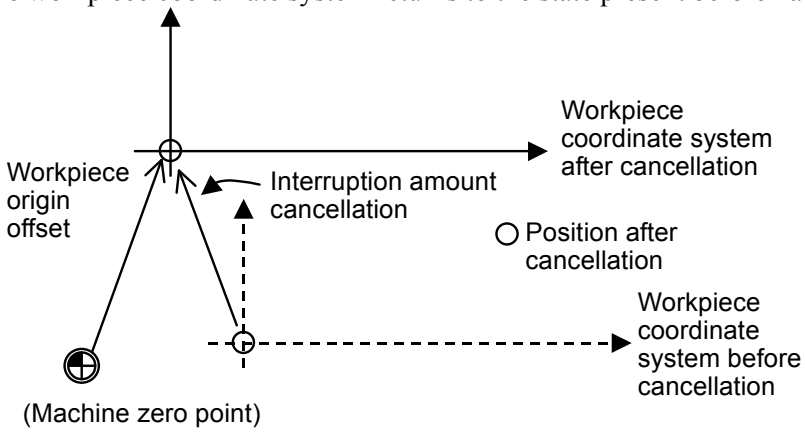
- Cancellation of the amount of interruption

Operation by which the workpiece coordinate system shifted by manual handle interruption from the machine coordinate system is returned to the original workpiece coordinate system is referred to cancellation of the amount of interruption.
When the amount of interruption is canceled, the workpiece coordinate system is shifted by the amount of manual handle interruption, and the amount of interruption is reflected in the absolute coordinates.

Interruption shifts the workpiece coordinate system from the machine coordinate system.



By cancellation, the workpiece coordinate system returns to the state present before handle interruption.



In the following cases, the amount of interruption is canceled:

- When a reset is made (when bit 1 (RTH) of parameter No. 7103 is set to 1)
- When emergency stop state is canceled (when bit 1 (RTH) of parameter No. 7103 is set to 1)
- When a manual reference position return operation is performed
  (when G28 is specified before a reference position is established)
- When a reference position is set without dogs
- When the workpiece coordinate system is preset

NOTE
When the amount of interruption is cleared using soft keys, only the indication of the amount of interruption becomes 0, and the workpiece coordinate system remains unchanged.

- Clearing the amount of interruption by soft keys

Clearing of the amount of interruption means that the indication of the amount of interruption by manual handle interruption is set to 0. The workpiece coordinate system does not change.
"Clearing all axes" or "Clearing any axis" is performed on the path for which the amount of manual handle interruption is indicated.
When bit 3 (HLC) of parameter No. 7100 is enabled, soft key [INTRPT CANCEL], which is used for this operation, appears. When HLC is disabled, soft key [INTRPT CANCEL] does not appear.

To select "Clearing all axes" or "Clearing any axis", follow the procedure below.

1 Press function key 

on the MDI unit.

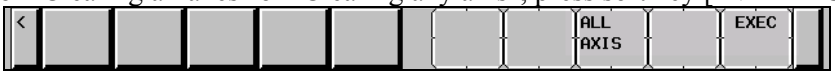
2 Press function key [HANDLE].



3 Press function key [(OPRT)].



4 To prepare for "Clearing all axes" or "Clearing any axis", press soft key [INTRPT CANCEL].



To prepare for "Clearing all axes" or "Clearing any axis", perform one of the following.

- Clearing all axes
  Press [INTRPT CANCEL] and then press [ALL AXIS].

- Clearing any axis
  - Enter the axis name and then press [INTRPT CANCEL].
  - Press [INTRPT CANCEL], enter the axis name, and press [EXEC].

If an incorrect axis name is entered, a warning message stating "FORMAT ERROR" appears.

- Relation with other functions

The following table indicates the relation between other functions and the movement by handle interruption.

Table 4.7 (a) Relation between other functions and the movement by handle interruption

Signals	Relation
Machine lock	Machine lock is effective. When machine lock is on, no movement is made due to handle interruption.

Signals	Relation
Interlock	Interlock is effective. When interlock is on, no movement is made due to handle interruption.
Mirror image	Mirror image is not effective. Interrupt functions on the plus direction by plus direction command, even if this signal turns on.

- Position display

The following table shows the relation between various position display data and the movement by handle interruption.

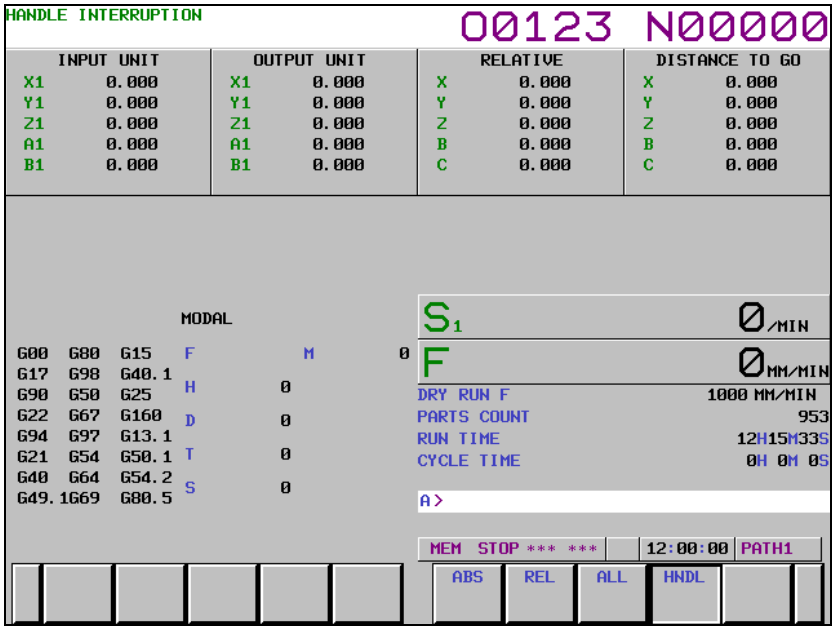
Table 4.7 (b) relation between various position display data and the movement by handle interruption

Signals	Relation
Absolute coordinate value	Handle interruption does not change absolute coordinates.
Relative coordinate value	Relative coordinates are changed by the travel distance specified by handle interruption.
Machine coordinate value	Machine coordinates are changed by the travel distance specified by handle interruption.

- Travel distance display

Press the function key 

, then press the chapter selection soft key [HNDL]. The move amount by the handle interruption is displayed. The following 4 kinds of data are displayed concurrently.



(a) INPUT UNIT:
Handle interruption move amount in input unit system
Indicates the travel distance specified by handle interruption according to the least input increment.

(b) OUTPUT UNIT :
Handle interruption move amount in output unit system
Indicates the travel distance specified by handle interruption according to the least command increment.

(c) RELATIVE:
Position in relative coordinate system
These values have no effect on the travel distance specified by handle interruption.

(d) DISTANCE TO GO:
The remaining travel distance in the current block has no effect on the travel distance specified by handle interruption.

The handle interruption move amount is cleared when the manual reference position return ends every axis.

- Display for five-axis systems or better

Systems having five or more axes provide the same display as the overall position display.

Note

NOTE

1 In a manual operation mode such as the jog feed mode, manual handle feed mode, or TEACH IN HANDLE mode, handle interruption cannot be performed.
2 During a machine lock or interlock, handle interruption does not cause movement.
3 Manual handle interruption is disabled for the axis in any of the following states.
- Follow-up state
- PMC axis control state
4 Manual handle interruption cannot be performed for the axis specified in the G00 mode.

4.7.1 Manual Interruption of 3-dimensional Coordinate System Conversion

Overview

When the manual pulse generator is rotated in the 3-dimensional coordinate conversion mode, the travel distance specified by the manual pulse generator is superposed on the travel distance by automatic operation in the direction of the selected handle feed axis on the coordinate (program coordinate) system after 3-dimensional coordinate conversion.

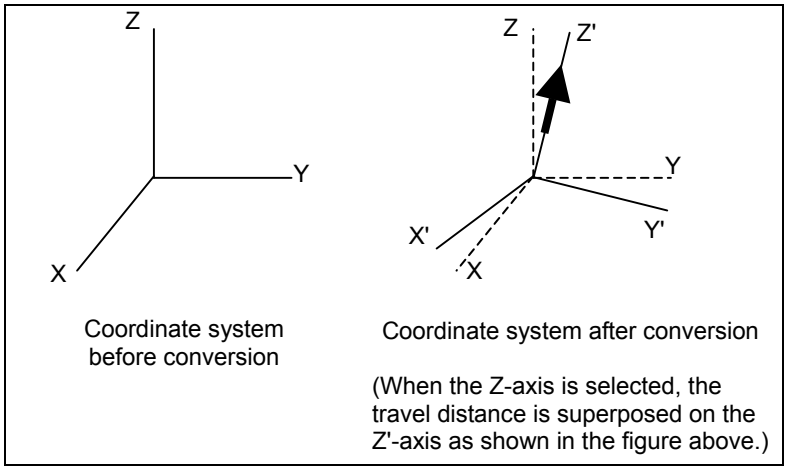


Fig. 4.7.1 (a) Manual interruption of 3-dimensional coordinate system conversion

Explanation

- Interruption

This function performs interruption only when all of conditions 1 to 6 below are met. Otherwise, interruption does not occur.

1. During the automatic operation mode (MEM mode, MDI mode, or DNC mode)
2. During the 3-dimensional coordinate conversion mode
3. During the cutting mode (G01, G02, G03, or other G code for cutting is being executed.)
4. When axis movement is performed along the three axes selected for 3-dimensional coordinate conversion
5. When 3-dimensional coordinate conversion manual interrupt enable/disable switch signal is set to 0

6. When the tool axis direction, the tool axis right-angle direction, or the tool tip center rotation is not selected for the 3-dimensional manual feed

The feedrate superposed along the 3-dimensional coordinate conversion mode does not exceed the maximum cutting speed of each axis.

- Manual interruption and coordinate system

When this function operates, the absolute coordinate values do not include the travel distance by manual interruption. The absolute coordinate values are therefore not updated even when the manual pulse generator is rotated.
The machine coordinate values and relative coordinate values include the travel distance by manual interruption. However, the values are represented on the coordinate system before being converted. As shown in Fig. 4.7.1 (a), the travel distance of the manual handle interruption is superposed along the Z'-axis. The travel distance is displayed using the X, Y, and Z coordinates of the coordinate system before being converted.
The travel distance by this function can be monitored on the manual handle interruption screen. Bit 3 (DMK) of parameter No. 5402 specifies whether the coordinate system (X, Y, Z) before being converted or the coordinate system (X', Y', Z') or after being converted is used to display the amount of interruption by the manual handle.

Note

Handle interruption is disabled during execution of a G68 or G69 block.

4.8 MIRROR IMAGE

During automatic operation, the mirror image function can be used for movement along an axis. To use this function, set the mirror image switch to ON on the machine operator's panel, or set the mirror image setting to ON from the MDI unit.

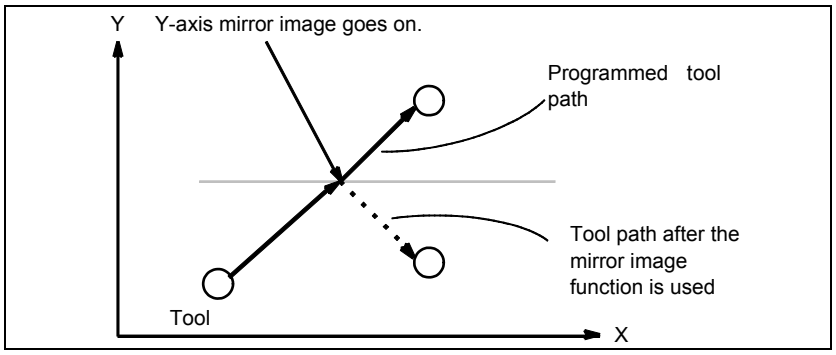


Fig. 4.8 (a) Mirror image

Procedure for mirror image

Procedure

The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

1 Press the single block switch to stop automatic operation.
When the mirror image function is used from the beginning of operation, this step is omitted.
2 Press the mirror image switch for the target axis on the machine operator's panel.
Alternatively, turn on the mirror image setting by following the steps below:
2-1 Set the MDI mode.
2-2 Press the function key .

2-3 Press the [SETING] soft key for chapter selection to display the setting screen.

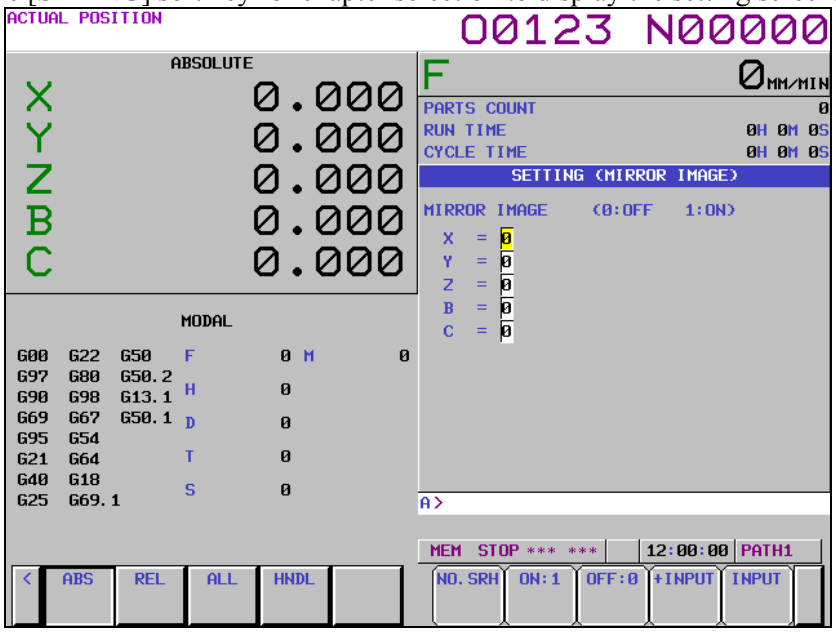


Fig. 4.8 (b) Setting screen

2-4 Move the cursor to the mirror image setting position, then set the target axis to 1.

3 Enter an automatic operation mode (MEM mode or MDI mode), then press the cycle start button to start automatic operation.

Explanation

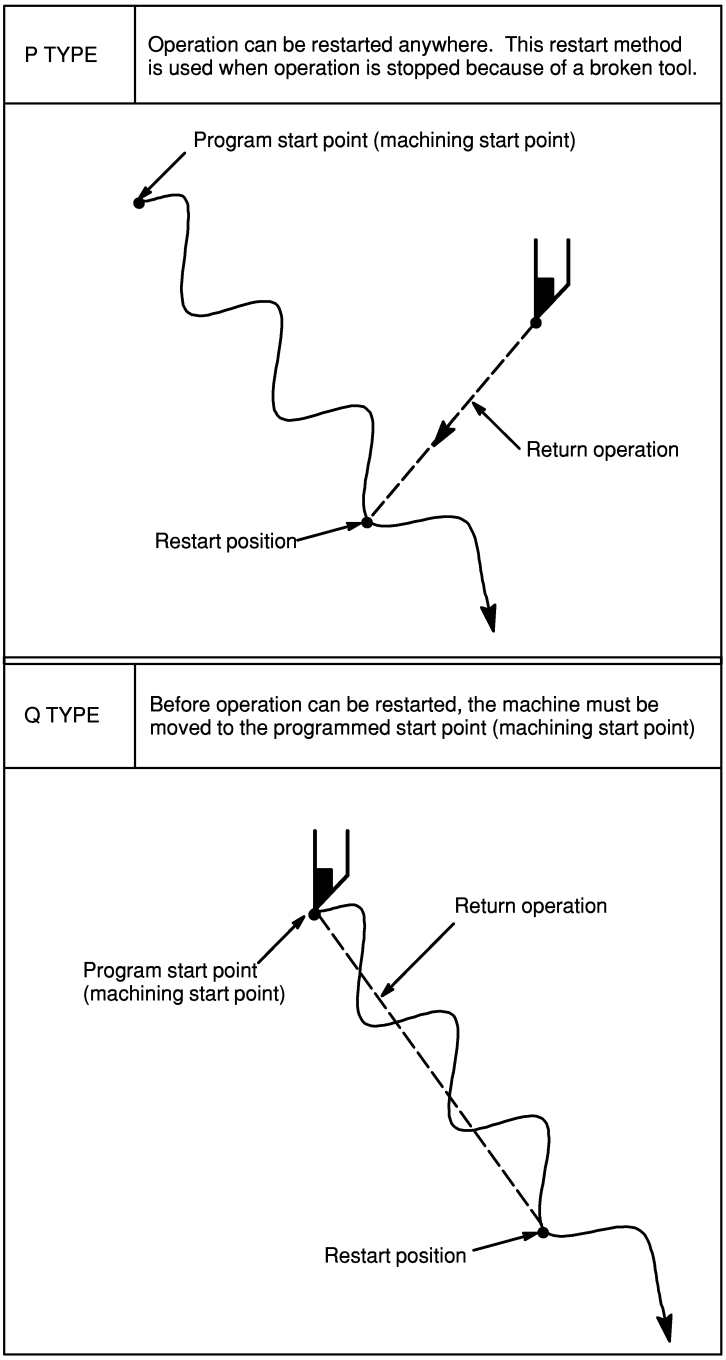
- The mirror image function can also be turned on and off by setting bit 0 (MIRx) of parameter No.0012 to 1 or 0.
- For the mirror image switches, refer to the manual supplied by the machine tool builder.

Limitation

The direction of movement during manual operation, the direction of movement from an intermediate point to the reference position during automatic reference position return (G28), the direction of approach during single direction positioning (G60), and the shift direction in a boring cycle (G76, G87) cannot be reversed.

4.9 PROGRAM RESTART

This function specifies Sequence No. of a block to be restarted when a tool is broken down or when it is desired to restart machining operation after a day off, and restarts the machining operation from that block. It can also be used as a high-speed program check function.
There are two restart methods: the P-type method and Q-type method.



Procedure for program restart by specifying a sequence number

Procedure 1

[P TYPE]

1 Retract the tool and replace it with a new one. When necessary, change the offset. (Go to step 2.)

[Q TYPE]

1 When power is turned ON or emergency stop is released, perform all necessary operations at that time, including the reference position return.
2 Move the machine manually to the program starting point (machining start point), and keep the modal data and coordinate system in the same conditions as at the machining start.
3 If necessary, modify the offset amount. (Go to step 2.)

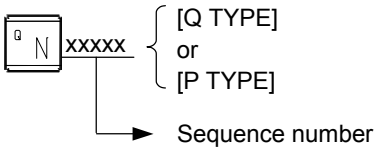
Procedure 2
[COMMON TO P TYPE / Q TYPE]

1 Turn the program restart switch on the machine operator's panel ON.

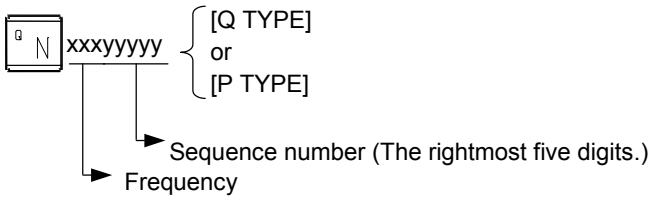
2 Press PROG key to display the desired program.

3 Find the program head. Press RESET key.

4 Enter the sequence number of the block to be restarted, then press the [P TYPE] or [Q TYPE] soft key.



If the same sequence number appears more than once, the location of the target block must be specified. Specify a frequency and a sequence number.



5 The sequence number is searched for, and the program restart screen appears on the LCD display.

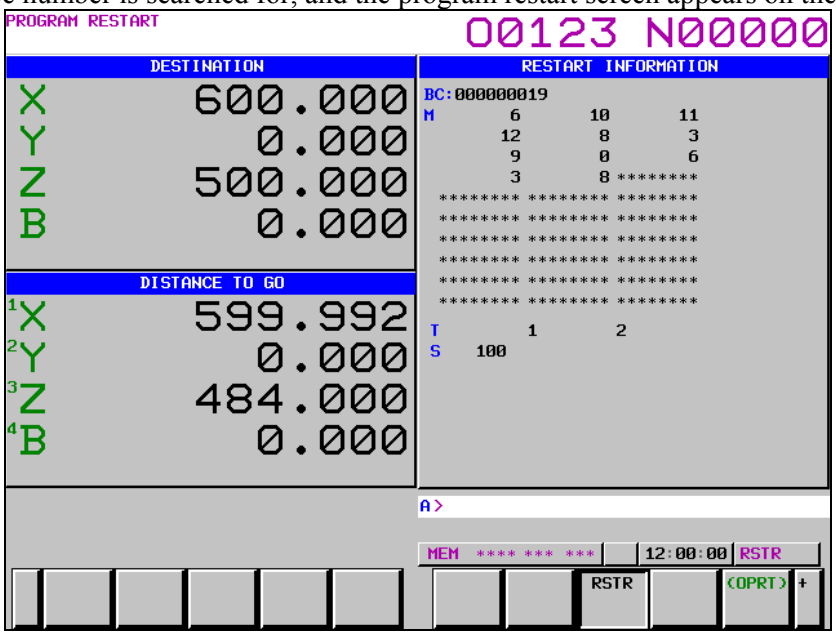


Fig. 4.9 (a) Program restart screen

DESTINATION shows the position at which machining is to restart.
DISTANCE TO GO shows the distance from the current tool position to the position where machining is to restart. A number to the left of each axis name indicates the order of axes (determined by parameter setting) along which the tool moves to the restart position.
The coordinates and amount of travel for restarting the program can be displayed for up to four axes. If your system supports five or more axes, pressing the [RSTR] soft key again displays the data for the fifth and subsequent axes.
M : Up to 35 most recently specified M codes. The maximum number of displayed M codes differs depending on the size of the display.

With 10.4-/15-/19-inch LCD/MDI unit : Up to 30 M codes
With 8.4-inch LCD/MDI unit : Up to 6 M codes

T : Two most recently specified T codes
S : Most recently specified S code
B : Most recently specified B code

Codes are displayed in the order in which they are specified. All codes are cleared by a program restart command or cycle start in the reset state.

6 Turn the program re-start switch OFF. At this time, the figure at the left side of axis name DISTANCE TO GO blinks.
7 Check the screen for the M, S, T, and B codes to be executed. If they are found, enter the MDI mode, then execute the M, S, T, and B functions. After execution, restore the previous mode. These codes are not displayed on the program restart screen.
8 Check that the distance indicated under DISTANCE TO GO is correct. Also check whether there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If such a possibility exists, move the tool manually to a position from which the tool can move to the machining restart position without encountering any obstacles.
9 Press the cycle start button. The tool moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified by parameter No. 7310 settings. Machining is then restarted.

Procedure for program restart by specifying a block Number

Procedure 1

[P TYPE]

1 Retract the tool and replace it with a new one. When necessary, change the offset. (Go to step 2.)

[Q TYPE]

1 When power is turned ON or emergency stop is released, perform all necessary operations at that time, including the reference position return.
2 Move the machine manually to the program starting point (machining start point), and keep the modal data and coordinate system in the same conditions as at the machining start.
3 If necessary, modify the offset amount. (Go to step 2.)

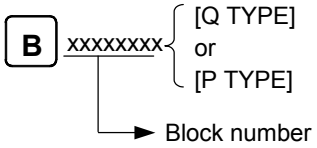
Procedure 2

[COMMON TO P TYPE / Q TYPE]

1 Turn the program restart switch on the machine operator's panel ON.
2 Press 

key to display the desired program.
3 Find the program head. Press 

key.
4 Enter the number of the block to be restarted then press the [P TYPE] or [Q TYPE] soft key. The block number cannot exceed eight digits.



5 The block number is searched for, and the program restart screen appears on the LCD display.

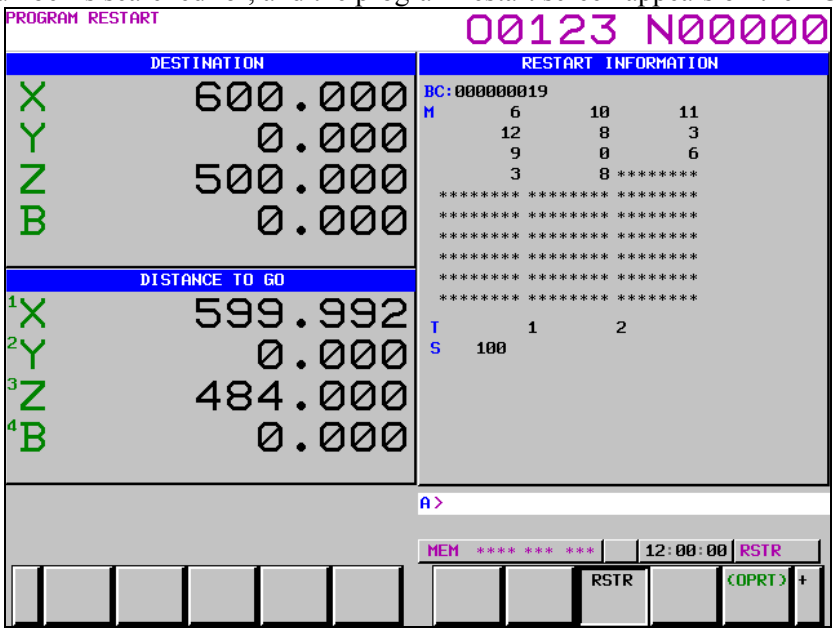


Fig. 4.9 (b) Program restart screen

DESTINATION shows the position at which machining is to restart.
DISTANCE TO GO shows the distance from the current tool position to the position where machining is to restart. A number to the left of each axis name indicates the order of axes (determined by parameter setting) along which the tool moves to the restart position.
The coordinates and amount of travel for restarting the program can be displayed for up to four axes. If your system supports five or more axes, pressing the [RSTR] soft key again displays the data for the fifth and subsequent axes.

M : Up to 35 most recently specified M codes. The maximum number of displayed M codes differs depending on the size of the display.
With 10.4-/15-/19-inch LCD/MDI unit : Up to 30 M codes
With 8.4-inch LCD/MDI unit : Up to 6 M codes

T : Two most recently specified T codes

S : Most recently specified S code

B : Most recently specified B code

Codes are displayed in the order in which they are specified. All codes are cleared by a program restart command or cycle start in the reset state.

6 Turn the program re-start switch OFF. At this time, the figure at the left side of axis name DISTANCE TO GO blinks.

7 Check the screen for the M, S, T, and B codes to be executed. If they are found, enter the MDI mode, then execute the M, S, T, and B functions. After execution, restore the previous mode. These codes are not displayed on the program restart screen.

8 Check that the distance indicated under DISTANCE TO GO is correct. Also check whether there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If such a possibility exists, move the tool manually to a position from which the tool can move to the machining restart position without encountering any obstacles.

9 Press the cycle start button. The tool moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified by parameter No. 7310 settings. Machining is then restarted.

Outputting the M, S, T, and B codes for program restart

After the block to be restarted is searched for, you can perform the following operations:

1 Before the tool is moved to the machining restart position
   <1> The most recently specified M, S, T, and B codes can automatically be output to the PMC. The most recently specified S code is output as the maximum spindle speed when the S code is specified in the block containing G92 or as the specified spindle speed in other cases. As the most recently specified S code, only one S code is displayed on the program restart screen regardless of whether the S code is specified in the block containing G92.
   <2> While the block to be restarted is being searched for, all sampled M codes and most recently specified S, T, and B codes can automatically be output to the PMC. Up to 35 M codes can be sampled. If the number of sampled M codes exceeds 35, the 35 most recently specified M codes are output to the PMC.
   Switch between operations <1> and <2> using bit 6 (MOA) of parameter No. 7300.
2 Before the tool reaches the machining restart position
   On the program restart screen, you can specify M, S, T, and B codes from the MDI unit in the MEM or DNC mode without changing the mode.

Outputting the most recently specified M, S, T, and B codes

When bit 7 (MOP) of parameter No. 7300 is set to 1, pressing the cycle start switch after searching for the block to be restarted automatically outputs the most recently specified M, S, T, and B codes to the PMC before the tool is moved to the machining restart position.
In the single block stop status, after the most recently specified M, S, T, and B codes are output, pressing the cycle start switch again moves the tool to the machining restart position.

Outputting all M codes and most recently specified S, T, and B codes

When bit 6 (MOA) of parameter No. 7300 is set to 1, pressing the cycle start switch after searching for the block to be restarted automatically outputs all M codes and most recently specified S, T, and B codes to the PMC before the tool is moved to the machining restart position.
(Example)
When M10, M11, M12, M13, M14, T0101, S1000, and B10 are sampled, a program is executed in the format shown below before the tool is moved to the machining restart position:

```
M10 T0101 S1000 B10 ;
M11 ;
M12 ;
M13 ;
M14 ;
```

Outputting M, S, T, and B codes on the program restart screen

When bit 7 (MOP) of parameter No. 7300 is set to 1, you can specify M, S, T, and B codes from the MDI unit in the MEM or DNC mode without changing the mode after searching for the block to be restarted until the tool reaches the machining restart position.

Procedure

1 When the block to be restarted is searched for using the program restart function, the program restart screen appears. When bit 7 (MOP) of parameter No. 7300 is set to 1, operation soft keys [OVERSTORE], [ERASE], and [INPUT] are displayed.

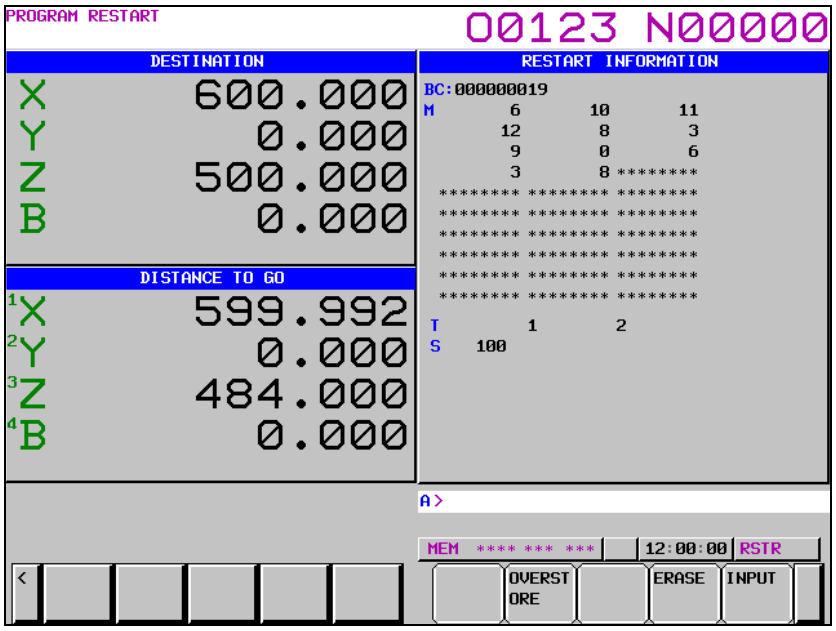


Fig. 4.9 (c) Program restart screen (outputting M, S, T, and B codes)

2 Before the tool reaches the machining restart position, pressing soft key [OVERSTORE] selects the over store mode. In the over store mode, data can be entered in the M, S, T, and B fields displayed in the (OVERSTORE) section.
To select the over store mode while the tool is moving to the machining restart position, hold restart operation by feed hold and press soft key [OVERSTORE].
Enter M, S, T, and B codes to be output in the (OVERSTORE) section from the MDI unit.
(Example)

To enter M10, S1000, T101, and B20 in the (OVERSTORE) section:

<1> Enter [M] [1] [0] from the MDI unit.

<2> Press the [INPUT] key.

You can also enter the S, T, and B codes by performing steps <1> and <2>.

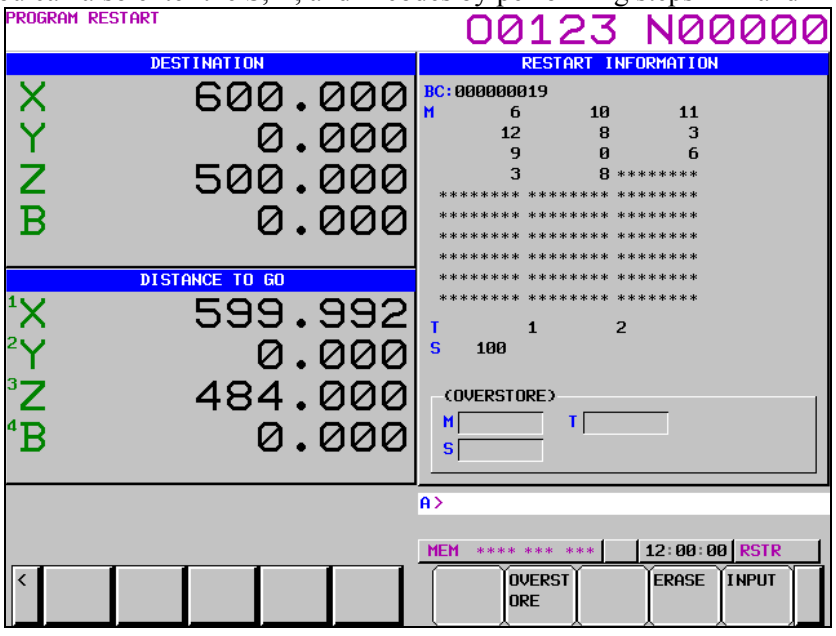


Fig. 4.9 (d) Program restart screen when M, S, T, and B codes are output

3 When values have been entered in the (OVERSTORE) section, pressing the cycle start switch outputs each code in the (OVERSTORE) section. The values in the (OVERSTORE) section are cleared.
4 To clear the values entered in the (OVERSTORE) section as M, S, T, and B codes, press soft key [ERASE]. All entered values are cleared.
5 Pressing soft key [OVERSTORE] again in the over store mode cancels the mode. Pressing the reset key also cancels the over store mode.
6 To continue with restart operation, cancel the over store mode and press the cycle start switch.

⚠ CAUTION
1 The M, S, T, and B codes specified in the over store mode are not displayed on the program restart screen.
2 In the over store mode, changing the operation mode to other than the MEM or DNC mode does not cancel the over store mode. In this case, no values can be entered in the (OVERSTORE) section.
3 In lathe systems, do not specify a T code in the over store mode. If a T code is specified, it is not executed.

Approach to the program restart position per arbitrary axis

This function achieves the following in a program restart:

- On the program restart screen, set the axis name of a single arbitrary axis by keying in or pressing a soft key. Then, perform a cycle start so that the target axis moves to the program restart position at the dry run speed, independently of the order set in parameter No. 7310.

To enable this function, set bit 7 (OAA) of parameter No. 11250 to 1.

Procedure

The procedure for a program restart by using this function is described below.

Enter the state in which a program restart is possible, as described below.
Enter either MEM or DNC mode.

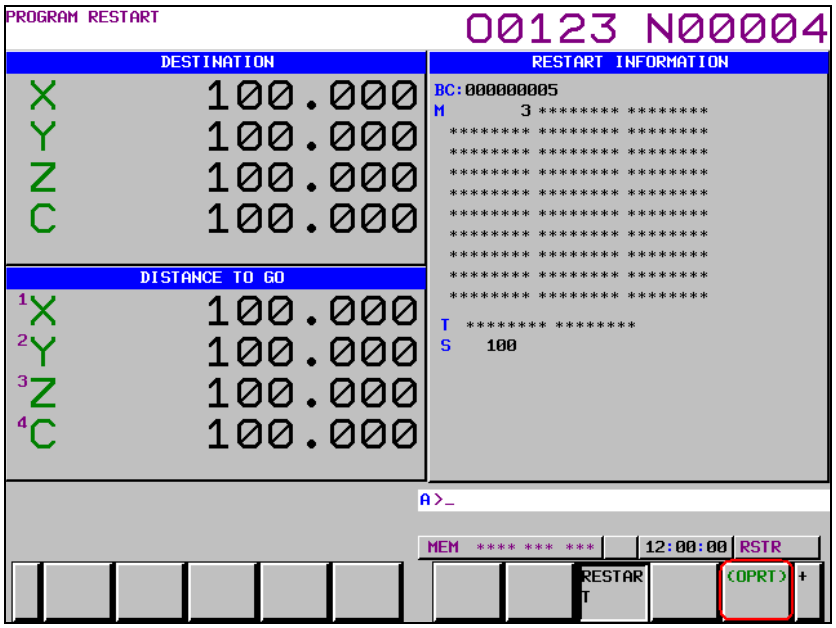


Fig. 4.9 (e) Program restart screen

Press the soft key [(OPRT)], then press the continuous menu key. Soft key [SET MV.AX] appears.

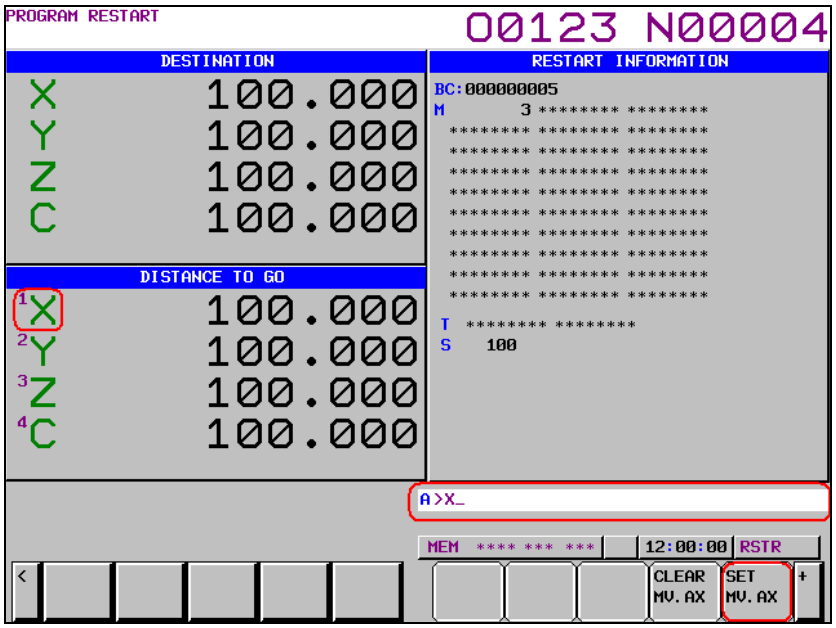


Fig. 4.9 (f) Program restart screen (movement axis setting)

Key in the axis name of the axis to move, and press the soft key [SET MV.AX], and the keyed-in axis name will flash.

NOTE
If an illegal axis is specified, the warning message, "THERE IS NO SPECIFIED NAME OF AXIS", appears.

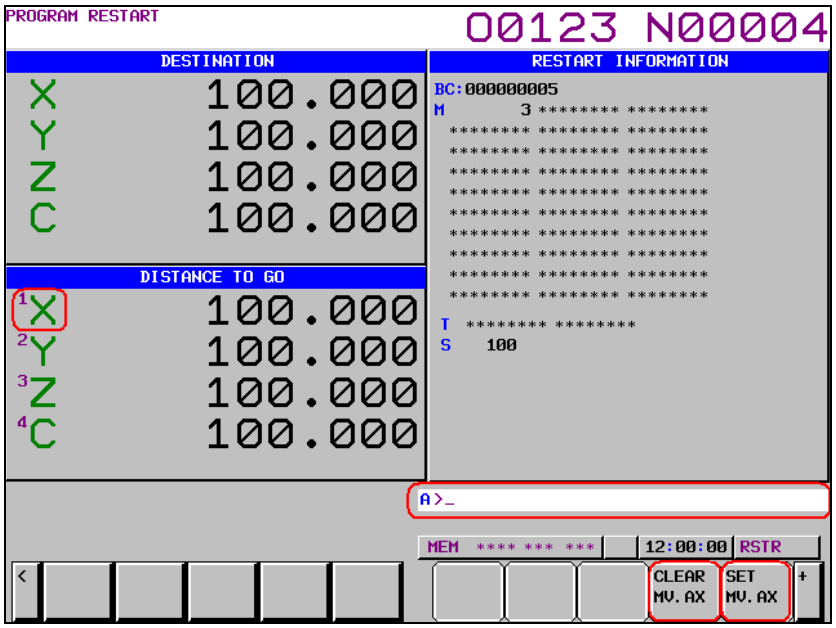


Fig. 4.9 (g) Program restart screen (movement axis setting, movement axis cancellation)

If wanting to cancel the axis that has been set and set another axis, key in this axis name, and press the soft key [SET MV.AX]. To move the axis, perform a cycle start, and the axis specified as described above will move to the program restart position at the dry run speed. When the movement of the axis starts, the axis name no longer flashes.

By pressing the soft key [CLEAR MV.AX], the axis that has been set as described above can be canceled. If the CNC mode is changed, the axis that has been set will be canceled (the axis name no longer flashes).

After the completion of the movement of all axes to the program restart position, perform a cycle start to restart machining.

If a cycle start is performed when the movement of all axes to the program restart position is not completed (or the axes are not yet moved) and no axis is set, the axes will move to the program restart position at the dry run speed. In this case, they will move in the order in which they are set in parameter No. 7310. Then, restart machining.

NOTE
1 If the soft key [CLEAR MV.AX] is pressed when no movement axis is set, the warning message, "THE MOVING AXIS HAS NOT BEEN SET", appears.
2 If the soft key [SET MV.AX] or [CLEAR MV.AX] is pressed in either of the states below, a warning message appears, making operation impossible.
- In a mode other than MEM and DNC modes
  Warning message: WRONG MODE
- Another axis is being moved, or restart coordinates are not set.
  Warning message: COMMAND ILLEGAL USE

Explanation

- Block number

When the CNC is stopped, the number of executed blocks is displayed on the program screen or program restart screen. The operator can specify the number of the block from which the program is to be restarted, by referencing the number displayed on the screen. The displayed number indicates the number of the block that was executed most recently. For example, to restart the program from the block at which execution stopped, specify the displayed number, plus one.
The number of blocks is counted from the start of machining, assuming one NC line of a CNC program to be one block.

(Example 1)

CNC Program	Number of blocks
O 0001 ;	1
G90 G92 X0 Y0 Z0 ;	2
G01 X100. F100 ;	3
G03 X01 -50. F50 ;	4
M30 ;	5

(Example 2)

CNC Program	Number of blocks
O 0001 ;	1
G90 G92 X0 Y0 Z0 ;	2
G90 G00 Z100. ;	3
G81 X100. Y0. Z120. R-80. F50. ;	4
#1=#1+1 ;	4
#2=#2+1 ;	4
#3=#3+1 ;	4
G00 X0 Z0 ;	5
M30 ;	6

Macro statements are not counted as blocks.

- Storing / clearing the block number

The block number is held in memory while no power is supplied. The number can be cleared by cycle start in the reset state.

- Block number when a program is halted or stopped

The program screen usually displays the number of the block currently being executed. When the execution of a block is completed, the CNC is reset, or the program is executed in single-block stop mode, the program screen displays the number of the program that was executed most recently. When a CNC program is halted or stopped by feed hold, reset, or single-block stop, the following block numbers are displayed:

Feed hold : Block being executed
Reset : Block executed most recently
Single-block stop : Block executed most recently

For example, when the CNC is reset during the execution of block 10, the displayed block number changes from 10 to 9.

- MDI intervention

When MDI intervention is performed while the program is stopped by single-block stop, the CNC commands used for intervention are not counted as a block.

- Block number exceeding eight digits

When the block number displayed on the program screen exceeds eight digits, the block number is reset to 0 and counting continues.

Program restart for flexible synchronous control

Parameter setting

By setting bit 0 (FRS) of parameter No. 13421 to 1, a program restart is possible in a program that contains an M code command for turning the flexible synchronous control mode on/off.
If bit 0 (FRS) of parameter No. 13421 is 0, and an attempt is made to perform a program restart in a program that contains an M code command for turning the flexible synchronous control mode on/off, alarm PS5244, “TOO MANY DI ON” is issued.

Creating/executing a program

1. If this function is used, it is not possible to specify a slave axis during flexible synchronous control. If an attempt is made to specify a slave axis, alarm PS5376, “FSC SLAVE AXIS CAN NOT COMMANDED” is issued.

   (Example)

   C: master axis, A: slave axis
   O0001 ;
   N10 C0 A0 ;
   N20 M133 ; ← Turns synchronous control mode on
   N30 C10. A5. ; → Alarm PS5376
   N40 C20. ;
   N50 M136 ; ← Turns synchronous control mode off
   N60 G90 A20. ;
   N70 G91 A10. ;
   :

2. When flexible synchronous control is canceled, a workpiece coordinate system preset is automatically performed on the slave axis.

3. If, after flexible synchronous control is canceled, an incremental command is issued for the slave axis before an absolute command, alarm PS5377, “INVALID COMMAND AFTER FSC OFF” is issued.

(Example)

C: master axis, A: slave axis

```
O0002 ;
N10 C0 A0 ;
N20 M133 ;          <-- Turns synchronous control mode on
N30 C10. ;
N40 C20. ;
N50 M136 ;
N60 G90 A20. ;      <-- Turns synchronous control mode off
N70 G91 A10. ;      <-- The slave axis requires an absolute command.
   :
```

Operating a program restart

1. Before starting a program restart, make sure that the flexible synchronous control mode matches the synchronous control mode on the program command in the block in which the restart is desired (state of the M code command for turning the flexible synchronous control mode on/off). If they do not match, alarm PS5374, “FSC MODE MISMATCH IN RESTART” is issued. In the period from the beginning of the program to the time the M code for turning the flexible synchronous control mode on/ff is issued for the first time, synchronous control mode is regarded as being off on the program command.

   (Example)

   C: master axis, A: slave axis

   ```
   O0003 ;
   N10 C90. ;
   N20 A90. ;
   N30 C0 A0 ;
   N40 M133 ;          <-- Turns svnchronous control mode on
   N50 C10. ;          <-- Restart block
   N60 C20. ;
   N70 M136 ;
   N80 G90 A20. ;      <-- Turns synchronous control mode off
   N90 G91 A10. ;
      :
   ```

   If the block in which to perform a restart is N50, it is necessary to enter the flexible synchronous control mode before performing a program restart.

2. During the execution of a program restart (period from the time the soft key [P TYPE] or [Q TYPE] is pressed until the end of the movement of all axes to the restart position), it is not possible to change the flexible synchronous control mode. If an attempt to change it, alarm PS5375, “FSC MODE CAN NOT CHANGED” is issued.

3. If a block under flexible synchronous control is specified as a restart block, the DISTANCE TO GO of the slave axis is 0. The restart position of the slave axis is "********".

4. If specifying a restart block after the cancellation of flexible synchronous control, specify a block after the absolute command for the axis that was the slave axis. If a block before the absolute command is specified, alarm PS5378, “INVALID RESTART BLOCK” is issued.

(Example)

C: master axis, A: slave axis
O0003 ;
N10 A90. ;
N20 C90. ;
N30 C0 A0 ;
N40 M133 ; ⟵ Turns synchronous control mode on
N50 C10. ;
N60 C20. ;
N70 M136 ; ⟵ Turns synchronous control mode off
N80 G90 A20. ;
N90 G91 A10. ;
:

If N80 is specified as a restart block, alarm PS5378 is issued.
If N90 is specified as a restart block, the alarm is not issued because N90 is after the absolute command for the slave axis of N80.

Continuing flexible synchronous control in the event of the emergency stop or servo-off state

Usually, if the machine enters the emergency stop or servo-off state, flexible synchronous control is canceled. By setting bit 1 (FCN) of parameter No. 13421 to 1, it is now possible to continue flexible synchronous control. (This function can be used even if bit 0 (FRS) of parameter No. 13421 is 0.)

In this case, if the machine enters the emergency stop or servo-off state, the difference between the current machine position of the slave axis and the correct synchronous control position in relation to the master axis becomes the positional deviation of the slave axis. If the emergency stop or servo-off state is canceled, the slave axis moves to the correct synchronous control position.

Reference position return/machine coordinate system selection during flexible synchronous control

Usually, if any of G27, G28, G29, G30, G30.1, and G53 is specified during flexible synchronous control, alarm PS0010, “IMPROPER G-CODE” is issued. If bit 2 (FRF) of parameter No. 13421 is set to 1, alarm PS5381, “INVALID COMMAND IN FSC MODE” is issued in either of the following cases. (This function can be used even if bit 0 (FRS) of parameter No. 13421 is 0.)

- G28 is issued for the master axis when the reference position is not established for the master axis under flexible synchronous control.
- If any of G27 to G30, G30.1, and G53 is issued for the slave axis.

Program restart for the Cs contour controlled axis

Usually, before a program restart operation can be started, it is necessary to enter Cs contour control mode and establish the origin of the Cs contour controlled axis.
By using the Cs axis coordinate establishment function and setting bit 5 (CCS) of parameter No. 7300 to 1, however, it is no longer necessary to enter Cs contour control mode or establish the origin of the Cs contour controlled axis before starting a program restart operation.
In this case, it is possible to change to Cs contour control mode, if necessary, after the end of the search for the block subject to the restart.
If the system enters Cs contour control mode, the origin of the Cs contour controlled axis is established with the Cs axis coordinate establishment function. And, the "DISTANCE TO GO" of the Cs contour controlled axis is recalculated.
If the system is not in Cs contour control mode, the "DISTANCE TO GO" of the Cs contour control axis is 0. "********" appears for "DESTINATION".

NOTE
To use the Cs contour controlled axis coordinate establishment function, the reference position return must be performed on the Cs contour controlled axis at least once after the power is turned on.

Program restart for 3-dimensional coordinate conversion

If the block subject to a restart is in 3-dimensional coordinate conversion mode G68 (machining center system)/G68.1 (lathe system), it is possible to select whether the movement of the axes to the restart position should be in the program coordinate system or the workpiece coordinate system, by using bit 1 (3DD) of parameter No. 7301. If parameter bit 3DD is "0", the axes move to the restart position one at a time, as seen from the program coordinate system, and if it is "1", they move to the restart position one at a time, as seen from the workpiece coordinate system.
The "DESTINATION" and "DISTANCE TO GO" displays follow the setting of 3DD.
The "ABSOLUTE" display follows the bit 6 (DAK) of parameter No. 3106.

Program restart for dynamic switching between diameter and radius

By using G10.9, it is possible to switch between diameter and radius dynamically and in a programmable way. G10.9 is effective even during a program restart.

Limitation

- P type Restart

In the following conditions, P type restart cannot be performed:

- Automatic operation has not been performed since power-on.
- Automatic operation has not been performed since emergency stop was released.
- Automatic operation has not been performed since the coordinate system was changed or shifted (change of the external workpiece origin offset value).

The block that can be restored properly by P type program restart is a block for which coordinate system setup or change was performed most recently before machining was interrupted.

- Restart block

The block where the program is to restart is not necessarily be the block at which the program was interrupted. You can restart the program from any block. For P-type restart, however, the block where the program is to restart must use the same coordinate system as when program execution was interrupted.

- Single block

When the single-block operation is enabled at the time of a movement to the restart point, a single-block stop occurs each time an axis operation takes place. In this case, no MDI operation is allowed.

- Manual intervention

During movement to the restart point, manual intervention is allowed for an axis for which a return operation has not yet been performed. However, manual operations do not cause any movement along axes for which a return operation has already been completed.

- MDI

When the search operation has ended, no move command can be specified by MDI before axis movement.

- Reset

Do not perform a reset operation during the time from the start of the search operation of the restart sequence until machining is restarted.
If a reset operation is performed, the restart steps must be performed again from the beginning.

- Feed hold

If a feed hold operation is performed during the search, the restart steps must be performed again from the beginning.

- Manual absolute

Every manual operation must be performed with the manual absolute mode turned on regardless of whether the manual operation is performed before or after machining.

- Reference position return

Unless an absolute position detector (absolute pulse coder) is provided, be sure to perform reference position return after power-up, then perform restart operation.

- Program restart switch

When the program restart switch is on, pressing the cycle start switch does not start operation.

- Blocks specifying a macro statement, macro call, and subprogram call

Blocks specifying a macro statement, macro call, and subprogram call are not searched for even when they have a sequence number. In such a case, search for a block previously preceding such a block.

- Interruption type custom macro

During movement to the machining restart point at a dry run feedrate, no interruption type custom macro can be started. If an interruption type custom macro is started, alarm DS0024, “UINT SIGNAL WAS ILLEGALLY INPUT” is issued.

- Index table indexing

When index table indexing is used for the machine, position the tool to the restart position before performing a program restart.

- Commands that prevent program restart

Program restart cannot be performed for blocks placed in the following modes:

- Polygon turning (G50.2)
- Threading (G32,G33), Circular threading (G35,G36), Threading cycle (G92), Multiple repetitive threading cycle (G76)
- Polar coordinate interpolation (G12.1)
- Balance cutting (G68)
- Rigid tapping

If any of the following commands is included between the beginning of a program and the block where the program is to restart, program restart cannot be performed:

- Workpiece coordinate system preset (G92.1,G50.3)
- Commands for enabling and disabling synchronous/composite control and superimposed control
- Commands for enabling and disabling axis synchronous control

- M, S, and T commands not usable in over store mode

The M, S, and T functions listed below, unlike the other M, S, and T functions, have special meanings within the CNC. These M, S, and T commands cannot be specified from the over store screen. To specify these commands, cancel the over store mode, and execute them in MDI operation.
Example:

- Spindle positioning
- Rigid tapping
- Tool management

Notes

(1) If bit 4 (INT) of parameter No. 13117 is 1, the interference check for cutter/tool nose radius compensation during a restart can be disabled.
(2) If bit 6 (SQB) of parameter No. 13117 is 1, a restart with a block number can be disabled. If B and a block number is specified as the block subject to a restart, a "format error" warning occurs.
If the actual execution of the conditional branch statement (if statement) of the custom macro program differs from its execution during a program restart, set parameter bit SQB to "1".
(3) If bit 7 (SQP) of parameter No. 13117 is 1, a P type restart is disabled. In this case, the soft key [P TYPE] no longer appears. If a P type program restart is not to be used, it is possible to disable the use of the P type by setting SQP to "1".
(4) A macro call block or a subprogram call block cannot be specified as the block subject to a restart.
(5) A tool direction control (G53.1) block cannot be specified as the block subject to a restart.
(6) The following system variables are not updated during a restart search:
#5021 to #5028 ABSMT
#5041 to #5048 ABSOT
#5061 to #5068 ABSKP

⚠ WARNING

As a rule, the tool cannot be returned to a correct position under the following conditions.
Special care must be taken in the following cases since none of them cause an alarm:

- Manual operation is performed when the manual absolute mode is OFF.
- Manual operation is performed when the machine is locked.
- When the mirror image is used. However, P type return is possible for a block that switched between ON and OFF most recently or a subsequent block. In this case, the mirror image signal status present when the program was interrupted must be maintained.
- When no coordinate system is set up at the beginning of a program in which main commands are executed in the incremental mode.
- When manual operation is performed in the course of axis movement for returning operation.
- When the program restart is commanded for a block between the block for skip cutting and subsequent absolute command block.
- When program restart is specified in the machine lock state, then the machine lock is canceled.
- When program restart specified for an intermediate block for a multiple repetitive canned cycle
- In general, when a coordinate system is set up, changed, or shifted after the search operation ends, the tool cannot be returned to a correct position.

⚠ CAUTION

Keep the following in mind when restarting a program including macro variables.

- Common variable
  When the program is restarted, the previous values are inherited as common variables without being preset automatically. Before restarting the program, initialize the appropriate variables to the original values used at start of the previous automatic operation.
- DI/DO
  At restart of the program, DI can be read by a system variable, but DO cannot be output.
- Clock
  When the program is being restarted, the clock time can be obtained by a system variable, but the time cannot be preset.
- Tool offset and workpiece origin offset
  When the program is being restarted, the offset can be read by a system variable, but change of the offset is allowed only for the Q type.

4.9.1 Auxiliary Function Output in Program Restart Function

Overview

This function provides the following features for program restart:

- M/S/T/B codes found during a search through a block to be restarted for operation are output to the program restart screen and an MDI program. Then the M/S/T/B functions can be executed from the MDI program.
- On a system having the M-code grouping feature, M codes are grouped. When they are output to the above MDI program, only the M code specified last, among the M codes in the same group, is output to the program restart screen and MDI program.
- M codes for calling subprograms/custom macros and their arguments are also output to the MDI program.
- The order in which individual axes move the tool to the machining restart position can be not only conventionally parameter-set but also set from the program restart screen.

Explanation

Program restart procedure

Follow the procedure below to restart a program using this function:

1. Turn on the program restart switch.
2. Display a program you want to restart and search for the beginning of the program.
3. Enter the sequence number (Nxxxx) or block number (Bxxxx) of a block you want to restart, then press the soft key [P TYPE] or [Q TYPE].
4. Upon completion of searching the block, the program restart screen will be displayed.
5. The M/S/T/B codes found by the search are output to the MDI program. (For details, see "M/S/T/B code output to the MDI program".)
6. On the program restart screen, set the order in which individual axes moves the tool to the machining restart position.
   (For details, see "Specifying the order in which axes move the tool to the machining restart position".)
7. Turn off the program restart switch.
   At this time, the number to the left of each axis name under "DISTANCE TO GO" blinks.
8. Switch to MDI mode and check the MDI program. Fix the MDI program as required, then execute cycle start.
9. When the execution of the MDI program is completed, return to MEM mode.

10. Check whether the distance in "DISTANCE TO GO" is correct and whether the tool does not hit the workpiece or any other object when moving to the machining restart position. If the tool hits anything, move the tool manually to a position free from interference.
11. Press the cycle start button. The tool moves on each axis in the order set in step 6 to the machining restart position at the dry run feedrate, then restarts machining.

NOTE
1 In steps 4, 7, and 11, the warning "ORDINAL NUMBER ERROR (RESTART)" is displayed if the order in which individual axes move the tool is set incorrectly.
2 If the warning occurs in step 4 or 7, you can perform the step that follows.
3 If the warning occurs in step 11, cycle start cannot be executed.
In this case, correct the order in which axes move the tool to the program restart position, then perform step 11 again.

M/S/T/B code output to the MDI program

This function causes the CNC to automatically output to the MDI program the M/S/T/B codes found during a search for a block in which to restart operation.

NOTE
1 Up to 50 M/S/T/B codes or macro call arguments can be output.
2 If there are no M/S/T/B codes to be output, the MDI program will not be created.
3 The number of characters that can be output to the MDI program is 512 including the program number (O0000) and EOB. If 512 characters are exceeded, only the latest command code of up to 512 characters is output.

- Codes that can be output to the MDI program

- S/T/B codes
  The code that appears last during a search is output. If, however, the T code is the same as the current modal T code, it is possible to specify whether to output it, by using bit 4 (MTO) of parameter No. 11250.

NOTE
1 In a lathe system, it is possible to specify whether to output T and B codes, using a parameter.
2 The S code of the clamp command of maximum spindle speed (G92S_; for the M series and G50S_; for the T series) is not output.

- M code
  For a system that has the M code grouping function (optional function), it is possible to group M codes from "M CODE GROUP" on the system screen.
  If grouped M codes are found, only the M code in the group that is specified last is output.
  All M codes that are not grouped are output. If, however, multiple M codes with the same number are specified, only the last single one is output.
  Subprogram call and custom macro call M codes are also output. Together with M codes, their arguments are also output to the MDI program. (For details, see "Output of subprogram/custom macro call M codes".)

NOTE
1 If multiple M codes are specified in a single block or if multiple MSTB codes are specified, it is possible to specify whether to output them to one block at a time or in the same block in the MDI program, by using bit 3 (MCO) of parameter No. 11250. (In either case, they are output in the MSTB order.)
2 If "0" or "1" is set as a group No., the M code is regarded as not belonging to any M code group.
3 Even if the option is enabled, it is possible to suppress a group check by setting bit 1 (MGC) of parameter No. 3400 to 1.

(Example)

Set N100 in the program O100 below as a restart block.
M code group settings
- M10, M11 : Group 2
- M120, M220, M320 : Group 3
- M51 : Group 0 (no group)

```
O100
M10                -- Group 2
 :
M11                -- Group 2
 :
M120               -- Group 3
 :
M51                -- Group 0
 :
M10                -- Group 2
 :
M220               -- Group 3
 :
M51                -- Group 0
 :
M320               -- Group 3
 :
```

M codes that are output

M10

M51

M320

Output of subprogram/custom macro call M codes

In the output of M/S/T/B codes to the MDI program, subprogram and custom macro call M codes are also output. Together with M codes, their arguments are also output to the MDI program.

NOTE
1 When the MDI program to which M/S/T/B codes are output is executed, the modal data and the workpiece coordinate system at the block position searched for during a program restart are effective. Thus, if a move command is to be executed in the program called from a custom macro call M code such as a tool change command, use the G53 (machine coordinate system selection) command so that it is not affected by the modal data or the workpiece coordinate system.
2 Those M/S/T/B codes that are specified in the subprogram and custom macro programs (including P-CODE calls) called with M codes are not output.
3 Waiting M codes are not output.
4 As a restart block, do not specify a block containing a subprogram call code.

- Subprogram call M codes

Subprogram call M codes specified with parameters Nos. 6044 to 6046 or Nos. 6071 to 6079 are output to the MDI program.

- Macro program call M codes

Macro program call M codes specified with parameters Nos. 6047 to 6049 or Nos.6080 to 6089 are output to the MDI program.

- Arguments of macro calls

Together with macro call M codes, their arguments are also output to the MDI program.

If the arguments of macro calls are to be output to the MDI program, and they are specified as macro variables or operations in the original program, the arguments are converted into numeric values.
(Example)

Original program	MDI program
#500 = 100. ; M20 A[#500/7] ;	M20 A 14.2857143 ;

NOTE
Conversion results are represented by 9-digit numbers, the decimal point, and the minus sign.
If a result cannot be represented by 9 digits, alarm PS5373, “ARGUMENT CONVERSION ERROR” is issued.
(Example)
In the example below, arguments cannot be converted into 9-digit numbers, so that alarm PS5373, “ARGUMENT CONVERSION ERROR” is issued.
M20 A[123456789.*10.] ; (1234567890. , 10 digits)
M20 B[0.00000001/10.] ; (0.000000001 , 10 digits)

Specification of the order of movement to the machining restart position

Usually, the order of movement to the machining restart position is the fixed one that is set in parameter No. 7310. This function enables changing of the order on the program restart screen when a program restart operation is performed.
The movement order can be changed after the end of the search for a program restart block until the start of the movement to the restart position.

- **Procedure for specifying the order of movement to the program restart position**

1. On the program restart screen, the cursor is displayed at the movement order display position to the left side of axis addresses.

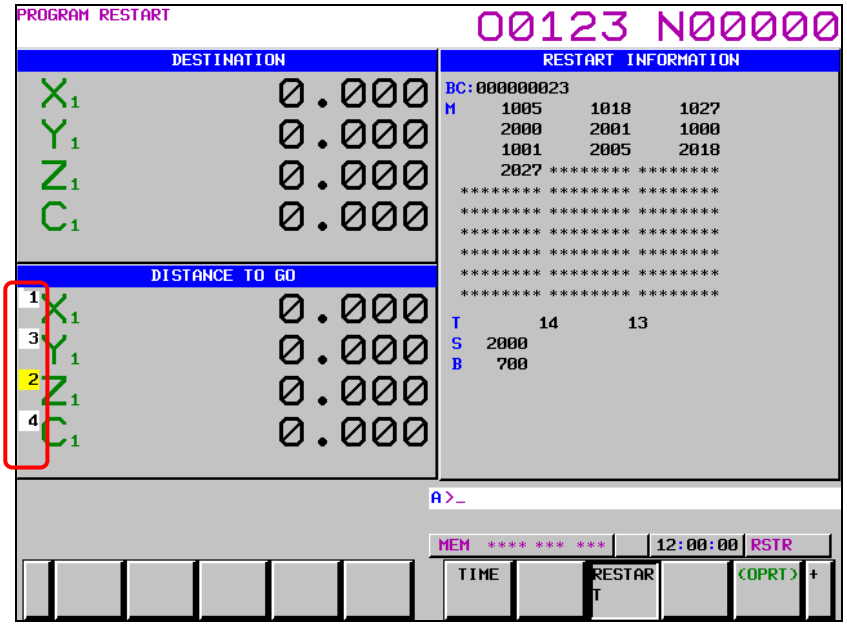


Fig. 4.9.1 (a) Program restart screen

2. Using the up and down arrow keys, position the cursor on an axis for which to set the movement order, and enter a numeric value.

NOTE
1 If the movement order is changed on the program restart screen, the value of parameter No. 7310 is also changed.
2 If an attempt is made to set a "0" or a value exceeding the number of controlled axes for an input value, the warning message, "DATA IS OUT OF RANGE", appears.

Note
If using this function, be sure to set bit 7 (MOU) of parameter No. 7300 to 0.

4.10 QUICK PROGRAM RESTART

This function restarts machining from a block by specifying the block on the relevant screen to restart machining after a tool is damaged or after holidays.

- For this function, a screen for restart operation is available. This screen is called “program restart setting screen”. To perform program restart, first specify a target block on the program restart setting screen, next press soft key [SEARCH EXEC]. Then, program restart is executed.
- On the program restart setting screen, information (including the program name and sequence number) on the last executed block is displayed as primary information. This information can be checked to see the machining interrupted point easily.
- On the program restart setting screen, in addition to the information on the last executed block, information on some blocks executed before that can be displayed. This information can be automatically stored in CNC memory during the operation of a program. This memory is called “program restart memory”.
  Block information stored in program restart memory is called “restart block information”.

- You can select an arbitrary block in restart block information and press soft key [SEARCH EXEC] to restart automatic operation from that block.

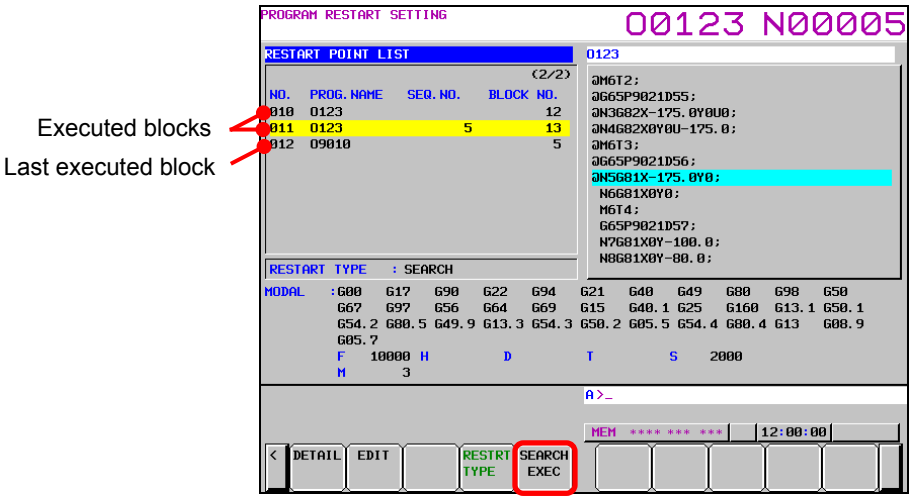


Fig. 4.10 (a) Program restart setting screen

- You can also restart a program after changing a block number or sequence number displayed on the program restart setting screen, using the MDI unit.
- To restore the machine to the suitable status (including the auxiliary function status and spindle status), use the "auxiliary function output in program restart" function. See Section 4.9.1, "Auxiliary Function Output in Program Restart Function" for details.
- For this function, either of the following two types of restart methods can be selected.

1. Search method

   Use this type when it is difficult to restore the status of modal information and auxiliary functions manually. This type can be selected to restart a program not only from the block stored in the program restart memory, but also from another block. When this method is selected, the modal information and position information of the block from which to restart the program can be restored automatically.

   Application cases : When the power is turned off after machining is interrupted
   When another workpiece is machined after machining is interrupted, and the restart block information on the previous workpiece is not stored
   When the program is to be restarted from a block which is not stored in the program restart memory

   Function : By using the "auxiliary function output in program restart" function, the auxiliary function status can be restored.
   The tool automatically moves to the restart position along each axis when the program is restarted.

   Operation : Restores modal information and coordinate values while performing the simulated execution of the program until the block specified in the restart block setting screen is found. (Updates modal information and coordinate values without moving the tool along any axis.)
   Mainly updates the following data:
   - Modal information including G, F, D, and H codes
   - Absolute coordinate system
   - Auxiliary function output in the program restart

2. Direct jump method

   Use this type when the operator restores the status of modal information and auxiliary functions manually and restarts the machining. This method can be selected only when the program is to be restarted from the block stored in the program restart memory.

When this method is selected, program restart is performed at high speed, though the modal information and position information of the restart block are not restored. Restart operation will be completed in a short time compared with the search method, in particular, when the restart block is in the middle of a long program.

Application cases : When the operator can easily restore the machine status because changes in the machine status are few when, for example, machining is interrupted, and is to be restarted immediately)
When a program is to be restarted from a block where operation can be started without restoring the machine status, such as a block on a machining process boundary

Function : The "auxiliary function output in program restart" function cannot be used.
The operator must determine whether to specify required auxiliary functions and specify them.
The machine must be positioned at the restart position manually before operation is restarted because the tool does not move to the restart position along any axis when the program is restarted.
Modal G codes and other modal information must be set to the suitable status by MDI operation before operation is restarted.

Operation : Moves only the program cursor to the specified block at high speed without restoring modal information or coordinate values.

- Executing cycle start after pressing soft key [SEARCH EXEC] on the program restart setting screen restarts operation. When the search method is selected, the auxiliary function status is restored by the "auxiliary function output in program restart" function and the tool is moved to the restart position before operation is restarted.

Procedure for restarting a program using the search method

Procedure 1

1. When power is turned on or emergency stop is released, perform all necessary operations at that time, including reference position return.
2. Confirm that the program to be restarted is selected. If you want to restart in the middle of a subprogram, confirm that the main program that calls the subprogram is selected.
3. Move the machine by manual operation to the program starting point (machining start point) and place the modal data and coordinate system in the same conditions as at the start of machining.
4. Modify the offset amount if necessary.
5. Go to procedure 2.

Procedure 2

1. Select the MEM or DNC mode.
2. Press function key 

. Press soft key [RESTRT], then [RESTRT SETING]. The following program restart setting screen appears.
The last item in the restart point list is the last block that was executed.
3. Go to procedure 3.

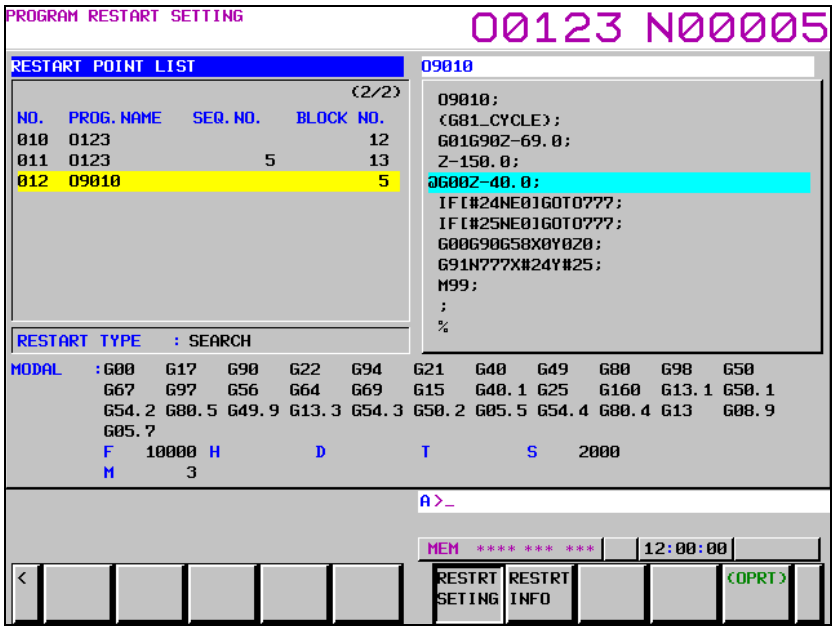


Fig. 4.10 (b) Program restart setting screen (restart point list)

NOTE

The program display shown on the right side of the screen in the figure above can be displayed only for the following cases:

(1) Memory operation
(2) Memory card program operation/editing
(3) Operation of a program in the Data Server as a program by memory operation or execution of the program by an M98 subprogram call

Accordingly, any program executed by an M198 external subprogram call or operated in the DNC mode is not displayed.

Procedure 3

1. Select a block from which to restart the program on the screen.
   To check detailed information of each block, press soft key, [(OPRT)], then [DETAIL]. Detailed information of the restart point pointed out by the cursor is displayed. To display the list screen again after checking detailed information, press soft key [LIST].
   - **When the block from which to restart the program is displayed**
     → Go to procedure 3-1.
   - **When a block near the block from which to restart the program is displayed**
     → Go to procedure 3-2.
   - **When neither of the above blocks is displayed**
     → Go to procedure 3-3.

Procedure 3-1 (When the block from which to restart the program is displayed)

1. Select a desired restart point using cursor keys   and page keys  .
   (Go to procedure 4.)

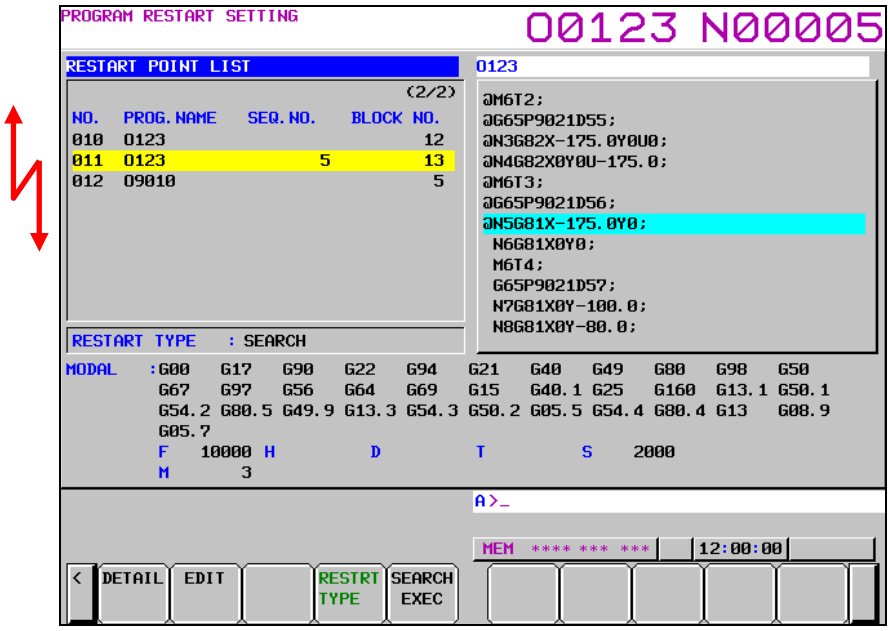


Fig. 4.10 (c)

Procedure 3-2 (When a block near the block from which to restart the program is displayed)

When a block near the block from which to restart the program is displayed as the restart point, specify the restart block by editing information of the restart point as follows:

1. Select a desired restart point using cursor keys   and page keys  .
2. Press soft key [(OPRT)], then [EDIT].
   Move the cursor and edit data.
   Enter desired sequence number, block number, and repetition count. When a sequence number is assigned to the restart block, specify the sequence number and repetition count without specifying any block number.

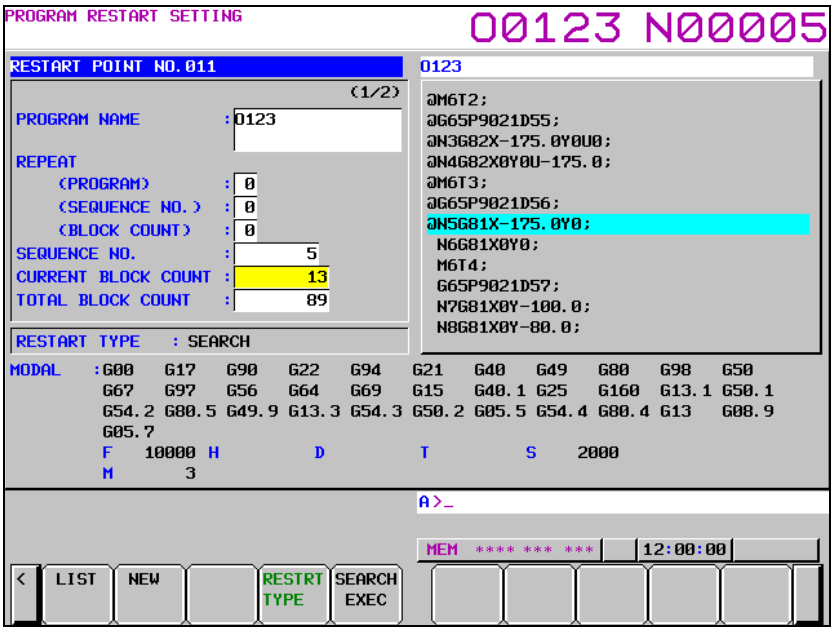


Fig. 4.10 (d) Program restart setting screen (edit)

NOTE

There are the following restrictions when two or more items are edited:

1 Among [SEQUENCE NO.], [CURRENT BLOCK COUNT], and [TOTAL BLOCK COUNT], only one item can be edited. Two or more items cannot be edited at a time. If one item is edited, the other two items become invalid.

2 In the [REPEAT] section, only one item can be edited. Two or more items cannot be edited at a time.
If one item is edited, the other two items become invalid.

The value in the [CURRENT BLOCK COUNT] field can be changed by selecting an arbitrary block in the program display window (on the right side of the screen) instead of entering the block number or sequence number. Follow the procedure below:

(1) Press the [→] key.

The cursor moves to the program display window (on the right side of the screen).

(2) Press the [↓] [↑] keys and select an arbitrary block from which to restart the program.

The value in the [CURRENT BLOCK COUNT] field in detailed information (on the left side of the screen) is automatically changed to the selected block.

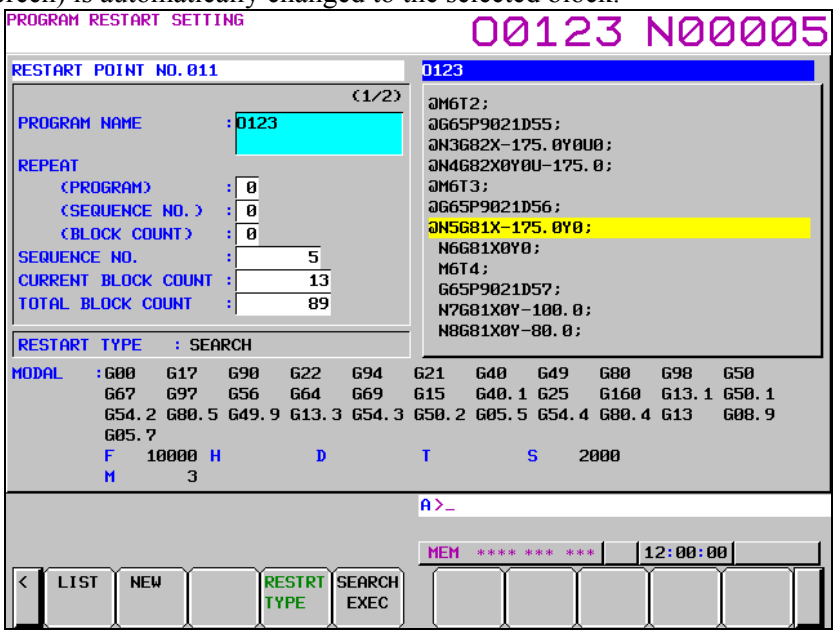


Fig. 4.10 (e) Specifying a block in the program display window

NOTE

This method is available for the following cases:

1. For restarting the program from a block in the memory operation mode
2. For restarting the program from a block in the memory card program operation/editing mode
3. For operating a program in the Data Server as a program by memory operation or executing it by an M98 subprogram call

Accordingly, this method is not available for a block in a program executed by an M198 external subprogram call or operated in the DNC mode.

3. Go to procedure 4.

Procedure 3-3 (When neither of the blocks is displayed)

Use this procedure when the block from which to restart the program is not displayed or if you want to restart another program that is not the interrupted program, from a desired block. Follow the procedure below:

1. Press soft key [(OPRT)] on the program restart setting screen. Then, press soft key [EDIT], then [NEW].
2. Enter the sequence number or block number, and other data of the block from which to restart the program to specify the restart block.

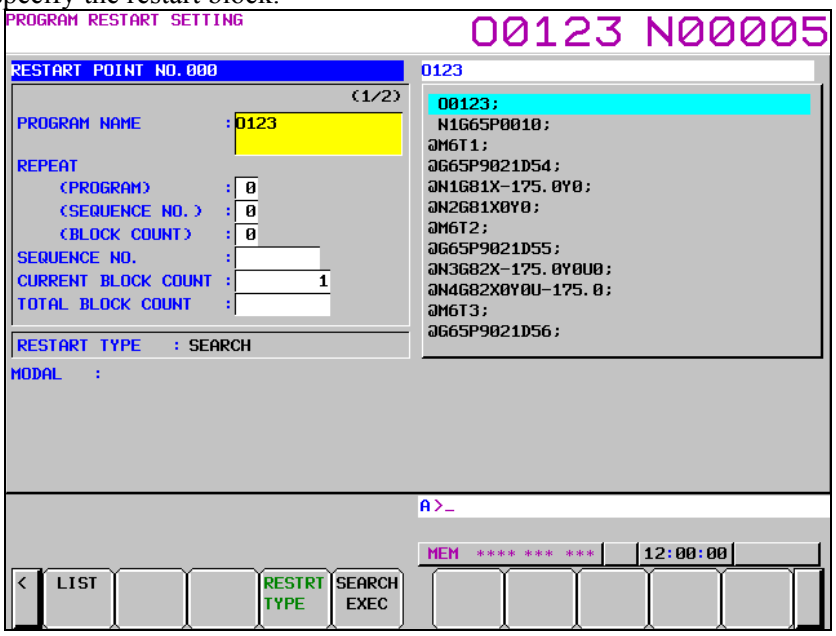


Fig. 4.10 (f)

Enter a value in the relevant field as follows:

(1) To specify a block with a sequence number
When a sequence number is assigned to the block from which to restart the program, set the sequence number in the [SEQUENCE NO.] field.
When the same sequence number appears in the program more than once, set the number corresponding to the block in the [REPEAT (SEQUENCE NO.)] field. If this number is omitted, the block is assumed to be the first block having the relevant sequence number that appears in the program.

(2) To specify a block with a block number
When no sequence number is assigned to the block from which to restart the program, set the number of that block in the [CURRENT BLOCK COUNT] field.
Specify the block number with the number of lines (EOBs) from the beginning of the program. When the same block is executed more than once according to the GOTO statement, specify the number of repetitions that corresponds to that block in the [REPEAT (BLOCK COUNT)] field. If this number is omitted, the block is assumed to be the first block.

(3) To specify a block with the number of execution blocks
When the number of blocks executed from the beginning of the program is known, set the number in the [TOTAL BLOCK COUNT] field.

3. Go to procedure 4.

Procedure 4

1. Press soft keys [(OPRT)], [RESTRT TYPE], and [SEARCH] to select the search method as the restart type. “SEARCH” is displayed for [RESTART TYPE] on the screen.

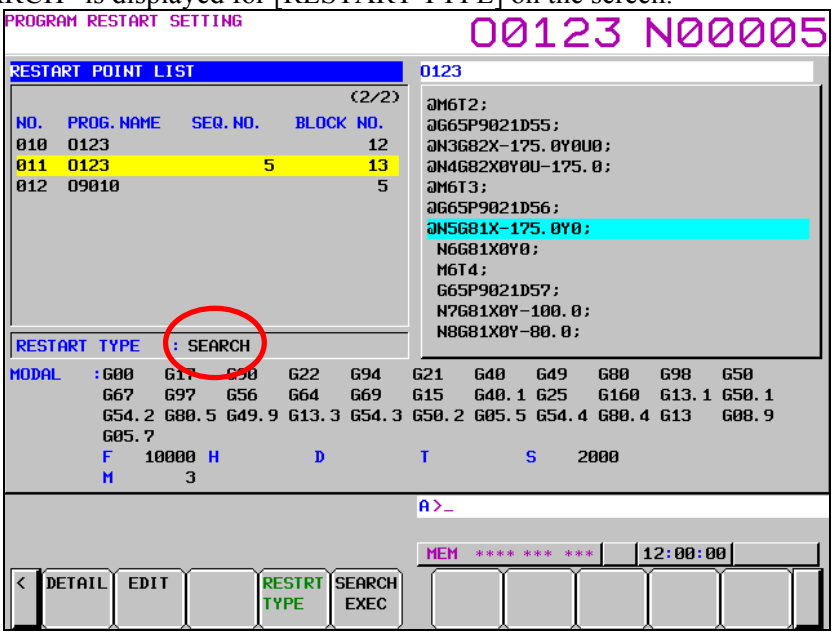


Fig. 4.10 (g)

NOTE
In the initial status after power-on, the search method is selected as the restart type.

2. Press soft key [SEARCH EXEC]. The selected restart point is searched for. When the search ends, the program restart screen appears.
   For details of this screen, see step 5 in “Procedure 2” in “Procedure for program restart by specifying a sequence number” in Section 4.9, “PROGRAM RESTART”.
3. Go to procedure 5.

Procedure 5

1. Set the order in which the tool moves to the machining restart position along axes on the program restart screen if necessary.
2. Set the MDI mode and check the MDI program. After modifying the MDI program if necessary, execute cycle start.
3. After the MDI program has been executed, return the machine to the MEM mode.
4. Check whether the distance displayed under [DISTANCE TO GO] is correct. Also check whether there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If such a possibility exists, move thee tool manually to a position from which the tool can move to the machining restart position without encountering any obstacles.
5. Press the cycle start button. The tool moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified in step 1.
   You can also move the tool to the machining restart position sequentially along axes by selecting an arbitrary axis at a time, independently of the order set in step 1. For details, see “Approach to the program restart position per arbitrary axis” in Section 4.9 “PROGRAM RESTART”.
   After the tool moves to the machining restart position along all axes, machining is restarted. Then, the program check screen automatically appears.

For details of steps 1 and 2 above, see Subsection 4.9.1, “Auxiliary Function Output in Program Restart Function”.

Procedure for restarting a program using the direct jump method

Procedure 1

1. When power is turned on or emergency stop is released, perform all necessary operations at that time, including reference position return.
2. Confirm that the program to be restarted is selected. If you want to restart in the middle of a subprogram, confirm that the main program that calls the subprogram is selected.

Procedure 2

1. Press function key 

. Press soft key [RESTRT], then [RESTRT SETING]. The following program restart setting screen appears.
   The last item in the restart point list is the last block that was executed.
2. Check whether the block from which to restart the program is displayed. If the block is not displayed, the program cannot be restarted using the direct jump method. Use the search method to restart the program.

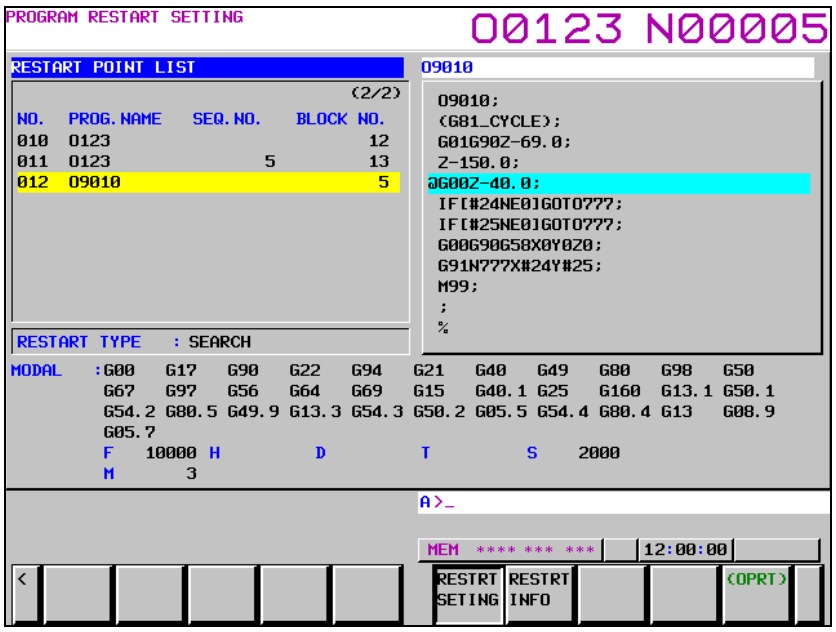


Fig. 4.10 (h)

3. Check the status of modal information and auxiliary functions in the block from which to restart the program. Restore the status by MDI operation if necessary.
4. Go to procedure 3.

Procedure 3

1. Select the MEM or DNC mode.
2. On the program restart setting screen, press soft keys [(OPRT)], [RESTRT TYPE], and [JUMP] to select the direct jump method as the restart type. "JUMP" is displayed for [RESTART TYPE] on the screen.

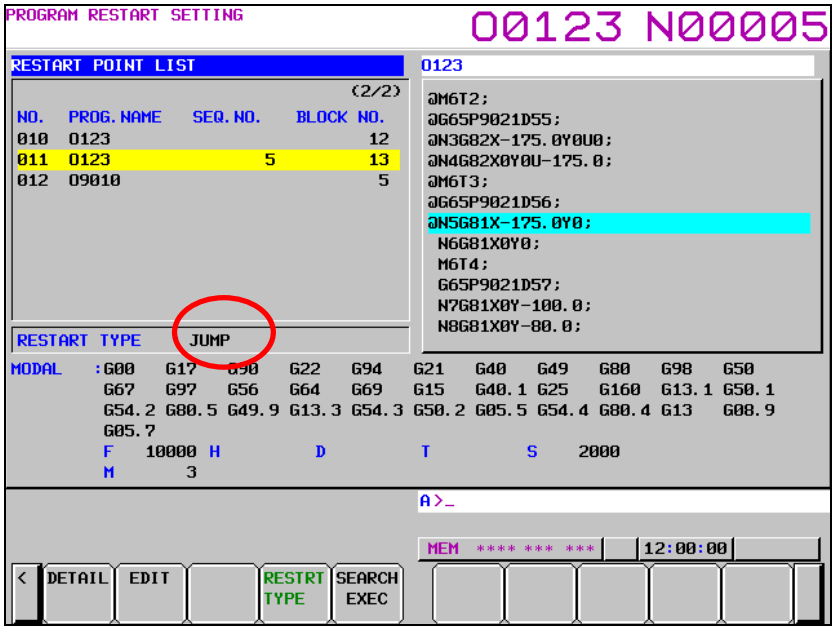


Fig. 4.10 (i)

3. Select the block from which to restart the program using the cursor and press soft key [SEARCH EXEC].
4. Go to procedure 4.

Procedure 4

1. The program cursor jumps to the specified block. On the program screen, check whether the cursor indicates the target block.
2. Check modal information and others again and specify the status by MDI operation if necessary.
3. Move the tool to the restart position by manual operation.
4. Return the machine to the MEM or DNC mode and execute cycle start.
   The program is executed from the cursor position.

Explanation

- Program restart memory

1. The following blocks are stored in the program restart memory:
   a. Last executed block
   b. Blocks of positioning
   c. Blocks of subprogram calls (including M98, M198, G65, G, and M code macro calls and subprogram calls)
   d. Other blocks judged to be suitable for restarting a program (blocks for turning each mode on or off and others that function as a breakpoint in a program)
2. If the program restart memory becomes full, block information is deleted from the older one and the information on a new block is overwritten.
3. The program restart memory is cleared when operation is started in the MEM or RMT mode after a reset.
   The memory is also cleared when the program is changed and restart operation using the search method starts.

4. When a program is restarted with a new restart point specified (“Procedure 3-3” in “Procedure for restarting a program using the search method”), the information on the blocks stored before restart operation is not cleared. After the program is restarted, block information after restart is added following the block information stored before restart.

- Program restart using the search method

1. Any sequence number in the programmable parameter input (G10L52 or G10L50) mode is not searched for when a search is executed with specifying a sequence number.
2. When a short cut is specified for a rotary axis, the tool also uses a short cut to the machining restart position.
3. If the single-block operation is enabled when the tool moves to the machining restart position, a single-block stop occurs each time the tool moves along an axis. During this operation, however, no MDI operation is allowed.
4. While the tool is moving to the machining restart position, manual intervention is allowed for an axis along which the tool has not moved to the restart position. The tool is not moved manually along any axis along which it has moved, however.
5. During movement to the machining restart position, no interruption type custom macro can be started. If an interruption type custom macro is started, alarm DS024 is issued.

- Procedure for restarting a program using the direct jump method

There are the following limitations.

1. A direct jump may not be able to be selected depending on the block. (Example: Block in the cutter or tool nose radius compensation mode)
   If an attempt is made to select “JUMP” as the restart type for such a block, the warning message, “ILLEGAL DATA”, is displayed and the setting is not accepted.
2. The direct jump method can be selected to restart a program from a block stored in the program restart memory. If an attempt is made to use the direct jump method to restart a program from a block which is not stored in the program restart memory, warning message “ILLEGAL DATA” is displayed.
3. When a direct jump method is selected, the status of items including position information, modal information, and macro variables is not restored. The “auxiliary function output in program restart” function is not also executed. For this reason, the operator must restore the status by manual or MDI operation with considering the status of the restart block.
4. In the following cases, a direct jump can be selected for a block in a subprogram or external subprogram call:
   a. Subprogram call using M98
   b. Subprogram call using G65
   c. Block in a subprogram on the Data Server operating by M198
   d. Block in a program on the Data Server operating in the DNC mode

   In other cases, a direct jump cannot be selected for a block in a subprogram or external subprogram call.

   That is, a direct jump cannot be selected for the following blocks:
   e. Block in a subprogram specified by an M or G code or a program called by a macro call
   f. Block operating in the DNC mode (other than the Data Server)
   g. Block in an execution macro
5. Note that there are the following limitations for blocks described in c. and d. above:
   (1) It is not possible to jump to the right position when there is a branch instruction of the custom macro.
   (2) It may take 1 to a few minutes to move the cursor to the restart block in a long program longer than 20 MB.
6. The jump position is shifted if the number of lines in the program is changed by editing the program after interrupted.

- Execution macro (macro executor)

A program cannot be restarted from a block in an execution macro.

- Modal display on the restart block setting screen

The modal display on the restart block setting screen indicates the status before the execution of the stored block. That is, data specified in the stored block is not displayed.

- Reset

Do not perform reset operation from the start of a search for restart to the restart of machining.
If reset operation is performed, perform restart operation from the beginning again.

- Feed hold

If performing feed hold operation during search, perform restart operation from the beginning again.

- Manual absolute

Perform every manual operation with the manual absolute mode turned on regardless of whether the manual operation is performed before or after machining.

- Reference position return

When no absolute position detector (absolute pulse coder) is provided, be sure to perform reference position return after power-on, then perform restart operation.

- Cs contour control

To restart a program containing a command for turning the Cs contour control mode on or off, whether the restart block is in the Cs contour control mode must be considered. Before starting restart operation, set the machine to the same mode as for the restart block.
However, by using Cs axis coordinate establishment, setting bit 5 (CCS) of parameter No. 7300 to 1, and the "auxiliary function output in program restart" function together, restart operation can be performed without considering whether the restart block is in the Cs contour control mode.
For details, see Item, "Program restart for the Cs contour controlled axis" in Section 4.9 "PROGRAM RESTART".

- Flexible synchronous control

Setting bit 0 (FRS) of parameter No. 13421 to 1 enables program restart in a program containing an M code for turning the flexible synchronous mode on or off.
For details, see Item, "Program restart for flexible synchronous control" in Section 4.9 "PROGRAM RESTART".

- Tilted working plane indexing

To restart a program from a block in the tilted working plane indexing mode using a direct jump method, follow the procedure below:

1. Return the tool to the restart position.
2. Execute [SEARCH EXEC] using the direct jump method.
3. Specify the tilted working plane indexing mode using the MDI unit after the program has been restarted (blinking RSTR disappears).
4. If the program is restarted from block after tool axis direction control, please command tool axis direction control.
5. The present absolute coordinate is matched to the absolute coordinates of the restart block.

- Tool center point control / High-speed Smooth TCP (Smooth control)

Execute a direct jump to the block in the tool center point control mode according to the following procedures.

1. Return the tool to the restart position.
2. Execute [SEARCH EXEC] using the direct jump method.

3. Specify the tool center point control mode using the MDI unit after the program has been restarted (blinking RSTR disappears).
4. The present absolute coordinate is matched to the absolute coordinates of the restart block.

Perform tool length compensation or tool offset operation by shifting the coordinate system (by setting bit 6 (TOS) of parameter No. 5006 to 1). And if direct jump restart is done except tool rotation type machine, please set the parameter INZ(No.19754#5) to 1. This parameter means the workpiece coordinate system is fixed to the rotary table, with the table rotation axis position being 0.

- Workpiece setting error compensation (or tilted working plane command + tool center point control)

When the search method is used

On a 5-axis machine with two rotary axes, one rotary axis functions to tilt the tool toward the workpiece. This rotary axis is referred to as a “rotary axis closer to the tool”. The other axis is referred to as a “rotary axis closer to the workpiece”. If the rotation direction error contains the error in the direction other than that of the rotary axis closer to the workpiece for workpiece setting error compensation, there are two types of combinations of the rotation angle solutions for tool direction compensation.

To restart a program from a block in the workpiece setting error compensation mode (or tool center point control mode in the tilted working plane indexing mode) using the search method, select one of the rotation angle solutions near the specified position (position of rotary axes before compensation).

When a direct jump method is used

Do not use the direct jump method to restart a program from a block in which the specified position of the rotary axis (position of the rotary axis before compensation) becomes the posture of the singularity.

To restart a program from a block in the workpiece setting error compensation mode (or tool center point control mode in the tilted working plane indexing mode), using the direct jump method, follow the procedure below:

1. Return the tool to the restart position.
2. Execute [SEARCH EXEC] using the direct jump method.
3. Specify the workpiece setting error compensation mode (or tool center point control mode in the tilted working plane indexing mode) using the MDI unit after the program has been restarted (blinking RSTR disappears).
4. The present absolute coordinate is matched to the absolute coordinates of the restart block.

- High-speed Smooth TCP (Rotation axes compensation)

To restart a program from a block in the rotation axes compensation mode of high-speed smooth TCP using a direct jump, follow the procedure below.

1. Return the tool to the restart position.
2 Execute a direct jump.
3. Specify the rotation axes compensation mode of high-speed smooth TCP using the MDI unit while the program restart mode (while blinking RSTR).
4 In the state of <3>, specify the rotation axes compensation mode of high-speed smooth (G43.4 L1).

Because a past value of the rotation axes compensation is not considered, there is a possibility that the coordinates are a little different.

Perform tool length compensation or tool offset operation by shifting the coordinate system (by setting bit 6 (TOS) of parameter No. 5006 to 1). And if direct jump restart is done except tool rotation type machine, please set the parameter INZ(No.19754#5) to 1. This parameter means the workpiece coordinate system is fixed to the rotary table, with the table rotation axis position being 0.

▪ Other notes

⚠ WARNING

In principle, the tool cannot return to a correct position in the following cases. Special care must be taken in the following cases since none of them cause an alarm.

- Manual operation is performed when the manual absolute mode is off.
- Manual operation is performed when the machine is locked.
- The mirror image is used.
- A program in which main commands are executed in the incremental mode does not begin with coordinate system setting.
- Manual intervention is performed while the tool is returning to the restart position along an axis.
- Program restart is specified for a block between the block for skip cutting and subsequent absolute command block.
- After program restart is specified in the machine lock state, the machine lock state is canceled.
- A program is restarted from a block in a multiple repetitive canned cycle with specifying a block or sequence number.
- In general, if a coordinate system is set up, changed, or shifted or a tool compensation value is changed, the tool cannot return to the correct position.

⚠ CAUTION

Keep the following in mind when restarting a program including a macro variable.

- Common variable
  When a program is restarted, the previous values are inherited as common variables without being preset automatically. For this reason, before restarting the program, initialize the required variables to the original values used at the start of the previous automatic operation.
- DI/DO
  During the restart of a program, DI can be read by a system variable, but DO cannot be output.
- Clock
  During the restart of a program, the clock time can be obtained by a system variable, but the time cannot be preset.

- Relation to other functions

The symbols in the table are described below:

A : Can restart a program from a block in the mode of each function.

NA : Cannot restart a program from a block in the mode of each function. Also cannot restart a program when any of these commands is specified in the first block of the program to the restart block.

*1 : There are some limitations. see Remarks.

*2 : Can restart a program after the status of items including modal information, macro variables, and auxiliary functions is restored by MDI operation or manual operation.

*3 : May restart a program using the “auxiliary function output in program restart” function in some cases.

*4 : Cannot restart a program from a block in the mode of each function.
Can restart a program from a block in which the mode is off when the program contains a block for turning the mode on or off.

*5 : Coordinate values are calculated on the assumption that the block is executed to the end point. Therefore, if the program is restarted from a block without specifying an absolute command at least once following this command, the positioned coordinate values may be different from those actually executed.

From blocks from which a program can be restarted using the conventional program restart function, a program can also be restarted using the search method.

Table 4.10 (a) Relation to other functions

Function name	Search method	Direct jump method	Remarks
Automatic operation (Memory operation)	A	A	
DNC operation	A	*1	Only the Data Server is available for a direct jump method.
DNC operation with memory card	A	NA	
Schedule function	NA	NA	
Optional block skip	*1	*1	Cannot be used when the signal is changed using an auxiliary function in the program.
Positioning	A	A	
Single direction positioning (G60)	A	A	
Linear interpolation	A	A	
Circular interpolation	A	A	
Helical interpolation	A	A	
Helical interpolation B	A	A	
3-dimensional circular interpolation	A	NA	
Dwell	A	A	
Involutes interpolation	A	A	
Exponential interpolation	A	A	
Conical/spiral interpolation	A	A	
Smooth interpolation	A	*4	
NURBS interpolation	*4	*4	
Nano smoothing	*4	*4	
Nano smoothing for 5-axis machining	*4	*4	
Thread cutting, synchronous cutting	*4	*4	
Multi threading	*4	*4	
Continuous threading	*4	*4	
Variable lead thread cutting	*4	*4	
Circular thread cutting	*4	*4	
Skip	*5	*2	
High-speed skip	*5	*2	
Multi-step skip	*5	*2	
Continuous high-speed skip	*5	*2	
Torque limit skip	*5	*2	
Polar coordinate interpolation	*4	*4	
Hypothetical axis interpolation	A	*4	
Cylindrical interpolation	*4	*4	
Polar coordinate command	A	*4	
Normal direction control	A	*4	
Gentle normal direction control	A	*4	
Mirror image	NA	*2	
Programmable mirror image	A	*4	
Mirror image for double turret	A	*4	
Scaling	A	*4	
Figure copying	A	*4	
Coordinate system rotation	A	*4	

Function name	Search method	Direct jump method	Remarks
3-dimensional coordinate system conversion	A	*4	
Tilted working plane indexing	A	*2	See item “Tilted working plane indexing” for details when a direct jump method is selected. See item “Workpiece setting error compensation (or tilted working plane indexing + tool center point control)” to use tool center point control simultaneously.
Workpiece setting error compensation	A	*2	See item “Workpiece setting error compensation (or tilted working plane indexing + tool center point control)” for details.
Rotary table dynamic fixture offset	A	*2	
Workpiece coordinate system	A	*2	
Workpiece coordinate system preset (G92.1/G50.3)	NA	*2	
Coordinate system setting (G92/G50)	A	*2	
Reference position return	A	A	
Reference position return check	A	A	
2nd reference position return	A	A	
3rd/4th reference position return	A	A	
Floating reference position return	A	A	
Tool length offset	A	*2	
Tool offset (G45 to G48)	A	*2	
Tool offset (T code on lathe system)	A	*2	
Tool radius / Tool nose radius compensation	A	*4	
Automatic tool nose radius compensation	A	*4	
Tool offset for Milling and Turning function	A	*2	
3-dimensional tool compensation	A	*4	
3-dimensional cutter compensation	A	*4	
Tool length compensation in tool axis direction	A	*4	
Tool center point control	A	*2	See item “Tool center point control” for details when a direct jump method is selected. See item “Workpiece setting error compensation (or tilted working plane indexing + tool center point control” to use the tilted working plane indexing simultaneously.
High-speed Smooth TCP (Rotation axes compensation)	A	*2	See item “High-speed Smooth TCP (Rotation axes compensation)” for details when a direct jump is selected. The direct jump used together with the tilted working plane indexing cannot be done.
High-speed Smooth TCP (Smooth control)	A	*2	See item ”Tool center point control” for details when a direct jump is selected.
Cutting point command	NA	*4	
Tool posture control	A	*2	
Automatic tool length measurement (M series) / Automatic tool offset (T series)	*5	*2	

Function name	Search method	Direct jump method	Remarks
Dynamic switching of diameter / radius specification	A	*2	
Plane selection	A	*2	
Balanced cutting	*4	*4	
Programmable data input	A	A	
Programmable parameter input	A	A	For the blocks between G10L52 (or G10L50) and G11, program restart with specifying a sequence number is disabled.
Sub program call (M98)	A	A	
Sub program call (M198)	A	*1	Only a program call on the Data Server is available for a direct jump method.
Inch/metric conversion	A	*2	
Direct drawing dimension programming	A	*4	
Chamfering/Corner R	A	*4	
Optional chamfering corner R	A	*4	
Custom macro	A	*2	
Interruption type custom macro	A	*2	
Canned cycle	*4	*4	
Multiple repetitive cycles	*4	*4	
Multiple repetitive cycles II	*4	*4	
Canned cycles for grinding	*4,*5	*4	
Canned cycle for drilling (except rigid tapping)	A	*4	
Small-hole peck drilling cycle	*4	*4	
Rigid tapping	*4	*4	
Axis control by PMC	*3	*2	
Real time custom macro	NA	*4	
Chopping	*3	*4	
Polygon turning	*4	*4	
Polygon turning by Cs contour control	*4	*4	
Polygon machining with two spindles	*4	*4	
High-speed cycle cutting	NA	*4,*2	
High-speed HRV3 control (G5.4)	*4	*2	
High-speed HRV4 control (G5.4)	*4	*2	
Optimum torque acceleration/deceleration	A	A	
Jerk control	A	A	
AI contour control I	A	A	
AI contour control II	A	A	
High-speed processing	A	A	
Exact stop mode	A	*2	
Tapping mode	A	*2	
Cutting mode	A	*2	
Automatic corner override	A	*2	
Auxiliary function	*3	*2	
2nd auxiliary function	*3	*2	
M code group check	*3	*2	
Tool management function	*3	*2	When the search method is selected and a tool exchange macro is executed after search, the amount of tool offset may be incorrect. (For a macro using a command such as H#8409D#8410)
Tool life management	*3	A	

Function name	Search method	Direct jump method	Remarks
Embedded macro	A	A	
Arbitrary angular axis control	A	A	
Axis synchronous control	NA	*4	Can be used when axis synchronous control is always on.
Parallel Axis Control	A	A	
Synchronous/Composite control	NA	*4	
Superimposed Control A	NA	*4	
Superimposed Control	NA	*4	
Synchronous/Composite/Superimposed control by program command	NA	*4	
Flexible synchronous control	*1	*1	See item “Flexible synchronous control”.
Hobbing/Electric gear box	NA	*4	
Electric gear box 2 pair	NA	*4	
Electric gear box automatic phase synchronization	NA	*4	
Spindle electric gear box	NA	*4	
Learning control	NA	*4,*2	
Preview repetitive control	NA	*4,*2	
Learning control for parts cutting	NA	*4,*2	
Learning Control for rigid Tapping	NA	*4,*2	
Machining condition selecting function	A	*2	
Path table operation	NA	*2	
Spindle positioning	*4	*4	The spindle must be positioned to the restart position by MDI operation in advance.
Cs contouring control	*3	*2	See item “Cs contour control” for details when the search method is selected.
Spindle speed fluctuation detection	A	*2	
Spindle synchronous control	*3	*2	
Live tool control with servo motor	*3	*2	
Spindle control with Cs contour control	*3	*2	
Servo/spindle synchronous control	*3	*2	
Macro executor	*4	*4	

4.10.1 Suppress Motion of Quick Program Restart

Overview

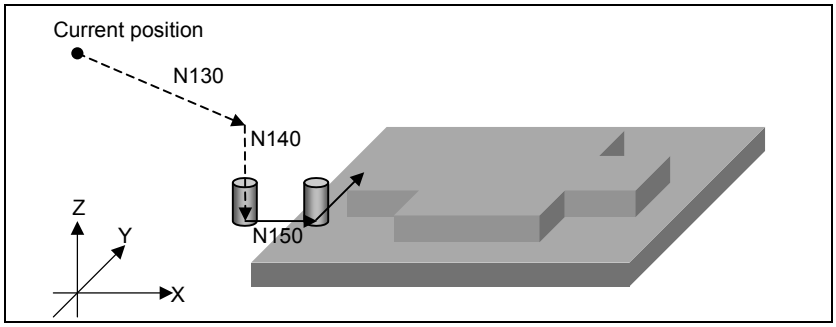
Suppress motion is a function which prohibits the approach to the start point of the specified restart block. In the conventional specification, the approach is performed to the start point of the restart block. To perform the approach to the end point of the restart block, the next block must be specified as the restart block. This function enables the tool to be moved to the end point of the specified restart block.
For example, when a block for approach is inserted in the machining program, the block for approach can be specified as the restart block and the tool can be moved to the end point of the block.

NOTE
An approach block is used for moving the tool closer to the workpiece from an arbitrary position (such as a tool change position). Normally, all axes used for the interrupted machining are specified. An approach block may consist of more than one block. In this case, select the first approach block as the restart block.

Example) Operation example

```
O1234;
:
N100 G91G28Z0.;
N110 G28X0.Y0.;
N120 T01M6;
N130 G90G00X-50.Y0;          Approach block for the X- and Y-axes
N140 Z-10.;                  Approach block for the Z-axis
N150 G41G01X-50.F1000.D1;
N160 Y50.;
:
```

When N130 is specified as the restart block, the tool moves from the current position to the end point of N130.

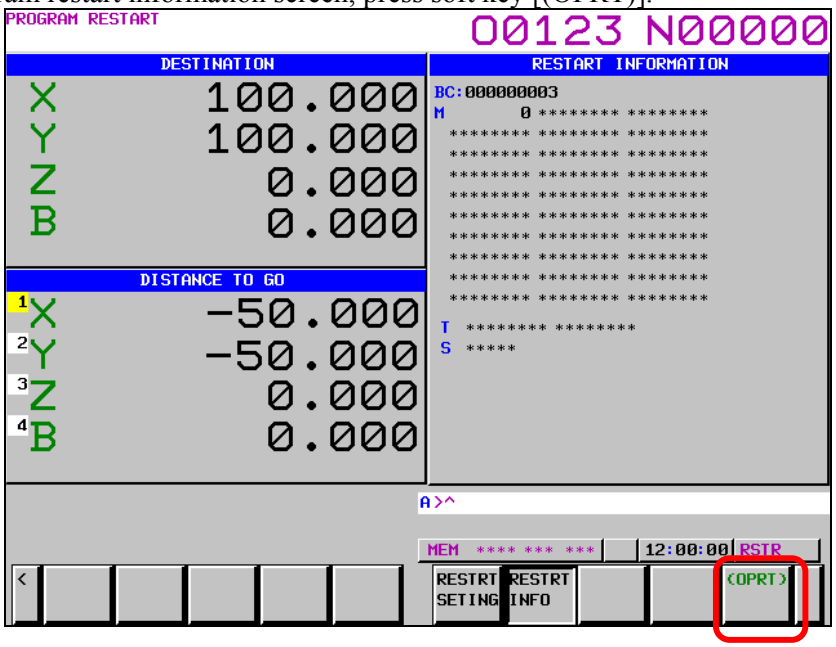


Explanation

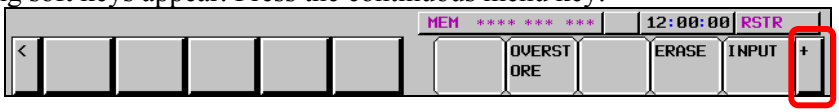
Setting bit 5 (SPR) of parameter No. 11250 to 1 enables suppress motion to be selected.

- Operation method

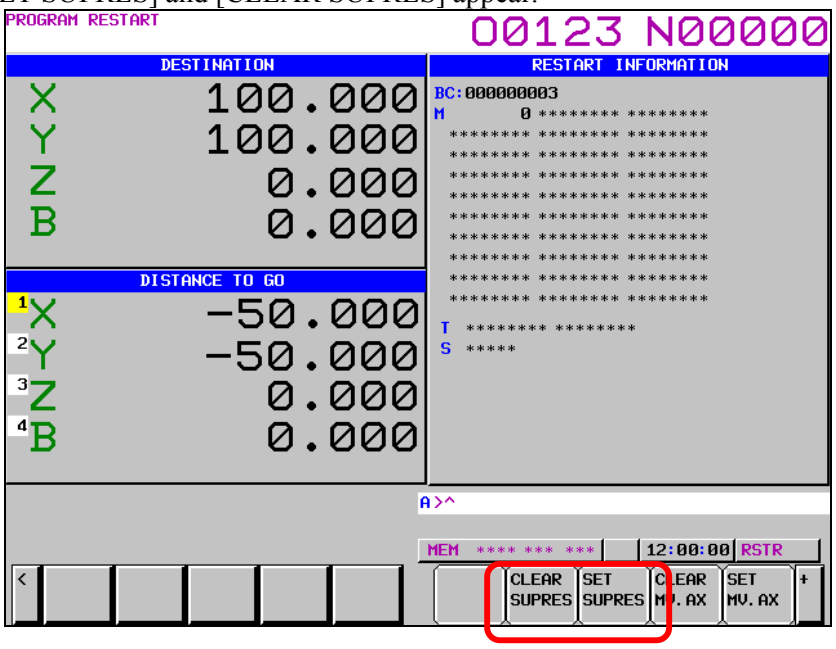
1. On the program restart information screen, press soft key [(OPRT)].



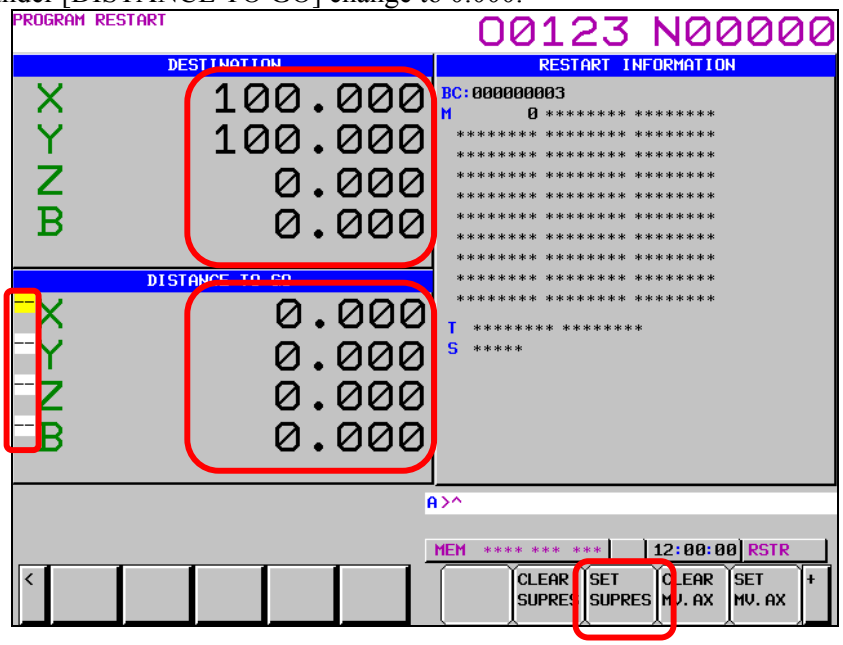
2. The following soft keys appear. Press the continuous menu key.



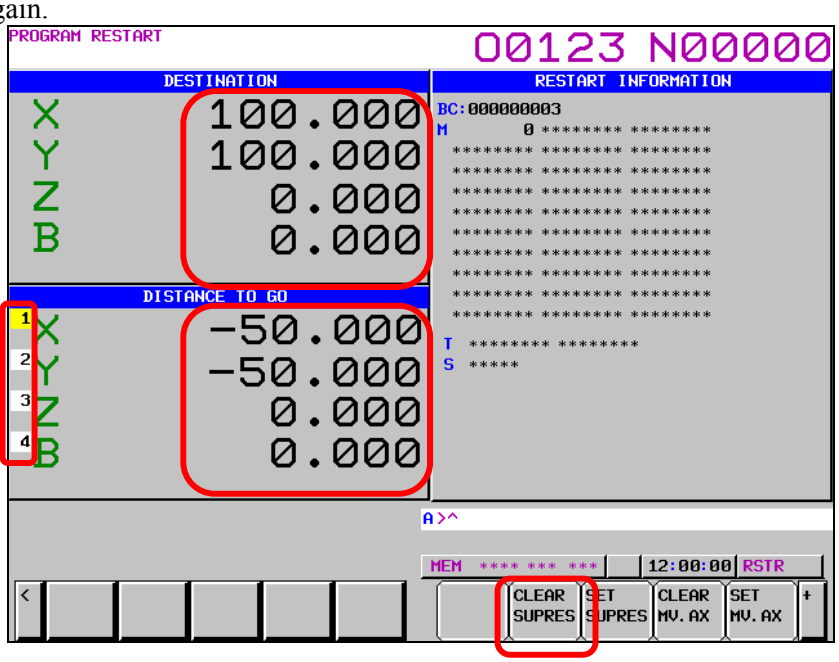
3. Soft keys [SET SUPRES] and [CLEAR SUPRES] appear.



4. To enable suppress motion, press soft key [SET SUPRES]. The number displayed on the left of each address changes to "--", the values under [DESTINATION] change to the current coordinates, and the values under [DISTANCE TO GO] change to 0.000.



5. To disable suppress motion, press soft key [CLEAR SUPRES]. The original numbers and values are displayed again.



- Block for which suppress motion is available

To perform suppress motion, select a block which satisfies conditions (1) to (3) below:

(1) An absolute command is specified in the block.
If an incremental command is specified in the restart block, alarm PS1930, “ILLEGAL COMMAND AFTER RESTART”, occurs.

(2) The G00 or G01 command is specified in the block.
If a command other than G00 or G01 is specified in the block, alarm PS1930 occurs.

(3) The block is not a modal block listed below.
If the restart block is a modal block listed below, suppress motion is not available even when the G00 or G01 command is specified in that block. If suppress motion is performed for a modal block listed below, alarm PS1931, “ILLEGAL MODE AFTER RESTART”, occurs.

- Programmable mirror image
- 3-dimensional coordinate system conversion
- Coordinate system rotation
- Tilted working plane indexing
- Tilted working plane indexing by tool axis direction
- Tilted working plane indexing (incremental multiple command)
- Scaling
- Polar coordinate interpolation
- Normal direction control
- Canned cycle for drilling
- Canned cycle

- Saving the suppress motion status

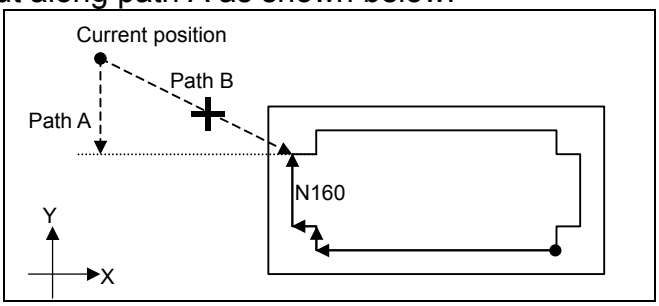
By setting bit 6 (SAV) of parameter No. 11250 to 1, the suppress motion status can be saved. With this function, once suppress motion setting is performed, the setting can be omitted after that.
The suppress motion status can be checked using parameter No. 7310. When suppress motion is used, a negative value is set in parameter No. 7310.
For an axis for which an invalid value is set in parameter No. 7310, the suppress motion status is not saved.

Caution

⚠ CAUTION

When suppress motion is used, the tool moves to the end point of the restart block only along the axis specified in the restart block. For this reason, the tool does not move to the end point of the restart block along any axis that is not specified in the restart block.

When N160 in the example is specified for the restart block, the tool moves not along path B, but along path A as shown below:



Along any axis that is not specified in the restart block, move the tool to a position where it will not hit anything when it moves during restart operation, in the manual mode in advance.

Note

NOTE

1 While the tool is moving along another axis, suppress motion cannot be set or canceled. If suppress motion is set or canceled while the tool is moving along another axis, the warning message, “COMMAND ILLEGAL USE”, is displayed.
2 Performing approach for each arbitrary axis in the suppress motion status cancels the suppress motion.

4.10.2 Quick Program Restart for a Machining Cycle

When the MANUAL GUIDE *i* function is enabled, program operation can be restarted in the middle of a hole machining cycle of MANUAL GUIDE *i*.

You can specify the restart position of a hole machining cycle on the detailed machining cycle restart information screen to restart a program easily.

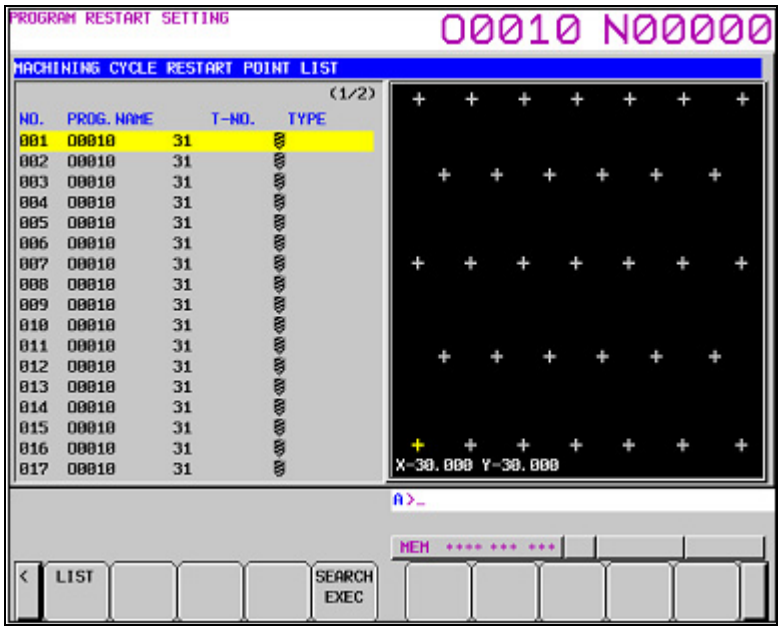


Fig. 4.10.2 (a) Detailed machining cycle restart information screen

NOTE
This function is available only for a hole machining cycle.

Procedure for quick program restart for a machining cycle

Procedure

The procedure for quick program restart for a hole machining cycle is described below:

1. In the MEM or DNC mode, execute the automatic operation of a hole machining cycle of MANUAL GUIDE *i*.

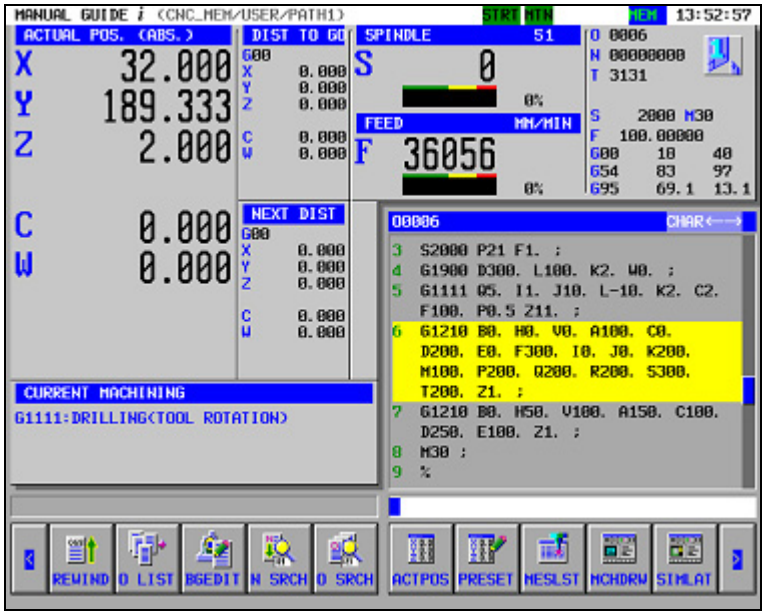


Fig. 4.10.2 (b) Base screen of MANUAL GUIDE *i* (1)

2. Interrupt or terminate the machining cycle, retract the tool, and replace it with a new one. Then, on the base screen, press soft key [RESTRT].

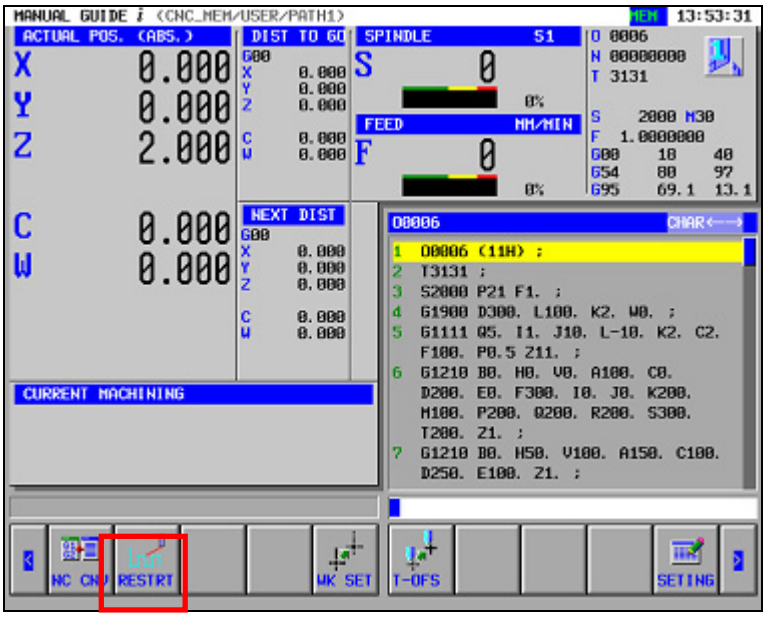


Fig. 4.10.2 (c) Base screen of MANUAL GUIDE *i* (2)

3. The CNC restart information list screen appears. Then, press soft key [MACHIN CYCLE].

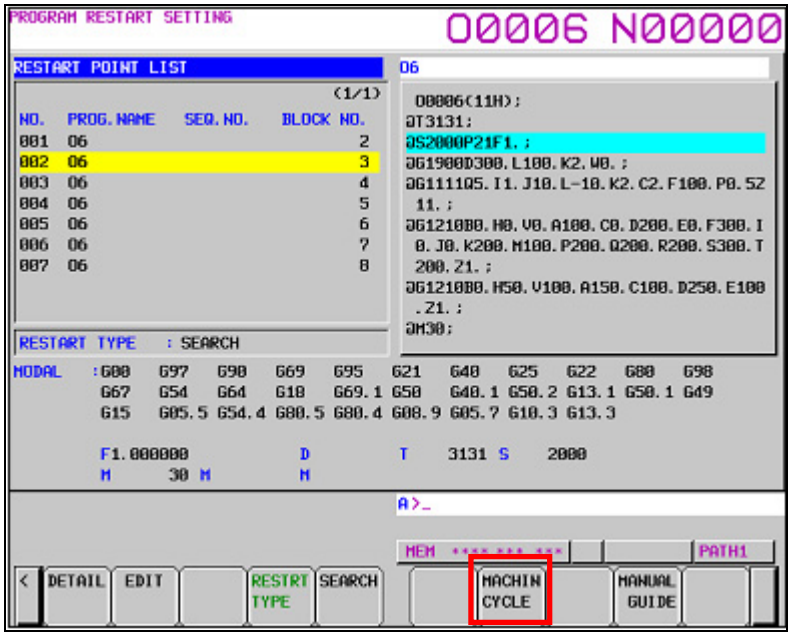


Fig. 4.10.2 (d) CNC restart information list screen

NOTE
Pressing soft key [MANUAL GUIDE] returns to the base screen of MANUAL GUIDE *i*. (when bit 2 of parameter No. 11351 is set to 1)

4. The CNC detailed machining cycle restart information screen appears. The screen displays a list of restart points (up to 50 points) for the hole machining cycle on the left side and a preview window for the positions of the holes (each hole position is indicated with “+”) on the right side. Move the cursor to the restart point of the hole position for restarting the cycle.

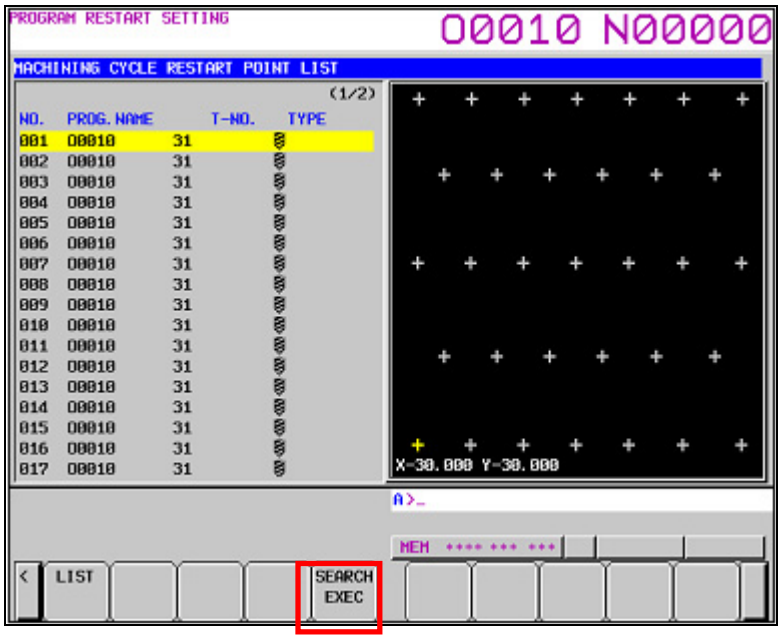


Fig. 4.10.2 (e) CNC detailed machining cycle restart information screen

5. Press soft key [SEARCH EXEC]. The machine takes a dry run to the restart point. At the same time, the program restart screen appears.

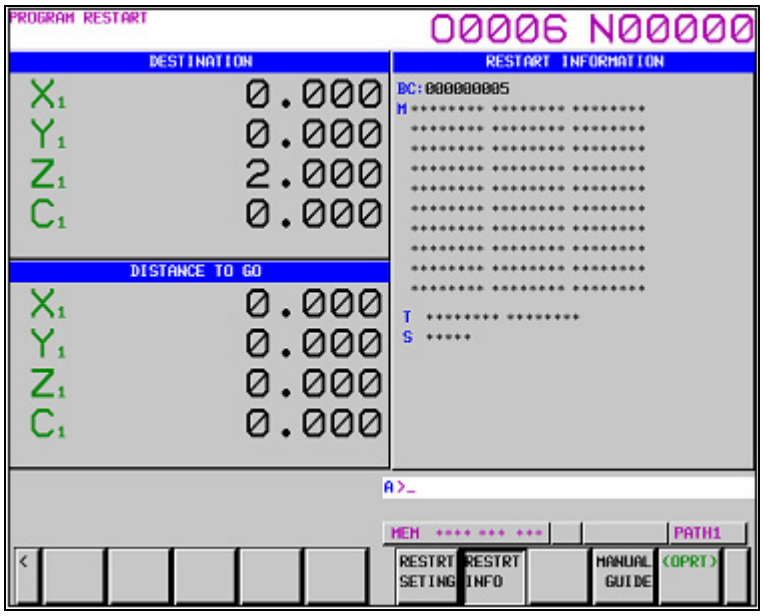


Fig. 4.10.2 (f) CNC program restart screen

NOTE
Pressing soft key [MANUAL GUIDE] returns to the base screen of MANUAL GUIDE *i*. (when bit 2 of parameter No. 11351 is set to 1)

6. Then, press the cycle start button. The tool moves from the current tool position to the restart point along axes in the order specified in parameter No. 7310 at the dry run feedrate. After the tool moves until the values under [DISTANCE TO GO] become 0, machining is performed from the restart point.

Explanation

- CNC detailed machining cycle restart information screen

When soft key [MACHIN CYCLE] is pressed on the CNC restart information list screen, the following screen appears:

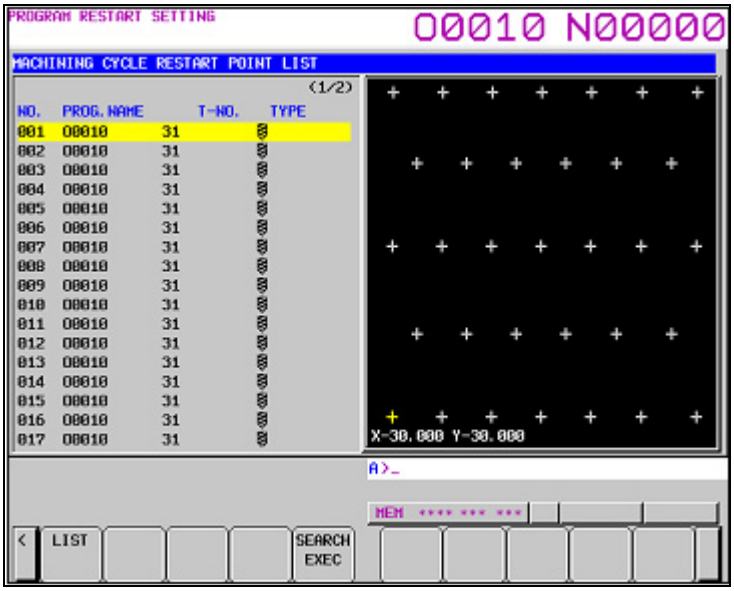


Fig. 4.10.2 (g) CNC detailed machining cycle restart information screen

The CNC detailed machining cycle restart information screen has the following soft keys.

Table 4.10.2 (a) Soft keys on the CNC detailed machining cycle restart information screen

Soft key	Description
LIST	Displays the CNC restart information list screen.
SEARCH EXEC	Performs a dry run to the restart point of the specified hole position and displays the CNC program restart screen.

This screen displays the following items.

Table 4.10.2 (b) Items displayed on the CNC detailed machining cycle restart information screen

Item	Description
NO.	Serial number assigned to each restart data item displayed in the list (up to three characters)
PROG. NAME	Execution program name (up to nine characters) If the program name consists of 10 or more characters, the first seven characters and "..." are displayed.
T-NO	T code or tool number (up to eight characters) (NOTE 1)
TYPE	Tool icon indicating a machining type. Each tool icon indicates a machining type as follows:  : Center drilling or drilling  : Boring, fine boring, or back boring  : Tapping or thread milling  : Reaming
Preview	"+" indicates the hole position of each machining profile. The hole position of the selected restart point is displayed in the cursor color (initial value: Yellow). The coordinates of the hole position are displayed at the bottom of the screen. (NOTE 2)

NOTE

1 The value displayed for T-NO differs depending on the settings of bit 2 of parameter No. 3108 and bit 0 of parameter No. 11320 as listed in the table below.

Table 4.10.2 (c) T-NO-related parameters

Bit 2 (PCT) of parameter No. 3108	Bit 2 (DHN) of parameter No. 11320	T-NO value
1	1	Spindle tool number (HD.T)
1	0	
0	1	
0	0	Modal T code When T0102 is specified in a lathe system, not compensation number 02, but tool number 1 is displayed. For a machine for which a tool in the standby position is specified, the number of the tool in the standby position is displayed for T-NO.

2 Color numbers specified on the CNC color setting screen are used for the preview display colors.

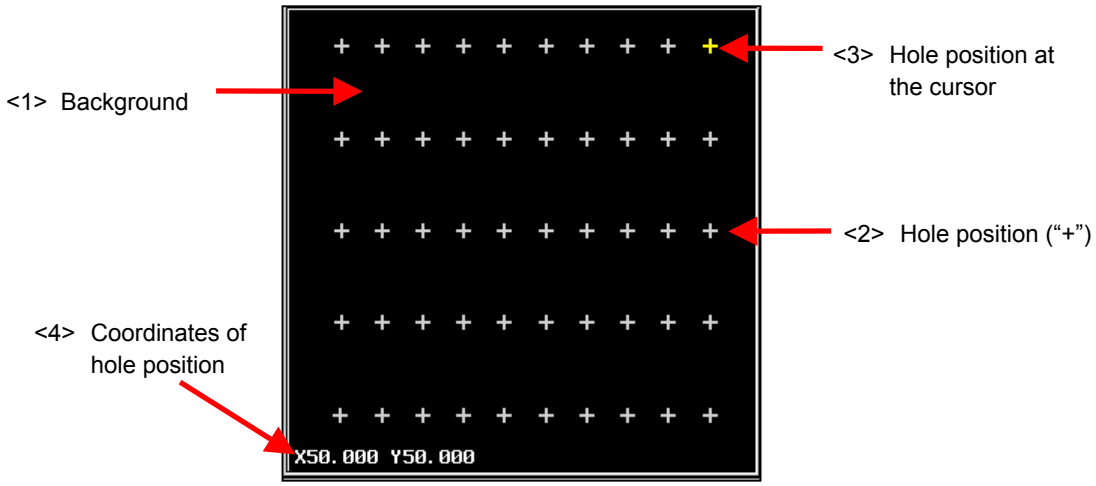


Fig. 4.10.2 (h) Preview screen configuration

Table 4.10.2 (d) Preview display colors

Preview display item	Color number	Initial value	Mainly used for
<1> Background	0	(fixed)	Soft key frame
<2> Hole position ("+")	15		Background color
<3> Hole position at the cursor	4		Cursor
<4> Coordinates of hole position	13		Alter possible data background

- Restart position of hole machining

The restart position of hole machining depends on "return mode (I)", input data of a hole machining cycle.

There are the following three return modes. The restart position is the block immediately before point R or I.

(1) [point I - 1]:
The tool returns to the point R level before moving from a hole to another hole. It returns to the point I level at the end of machining.
For the first hole position, machining restarts from the block immediately before point I (start point of operation 1). For the second or subsequent hole position, machining restarts from the hole position immediately before point R (start point of operation 5).

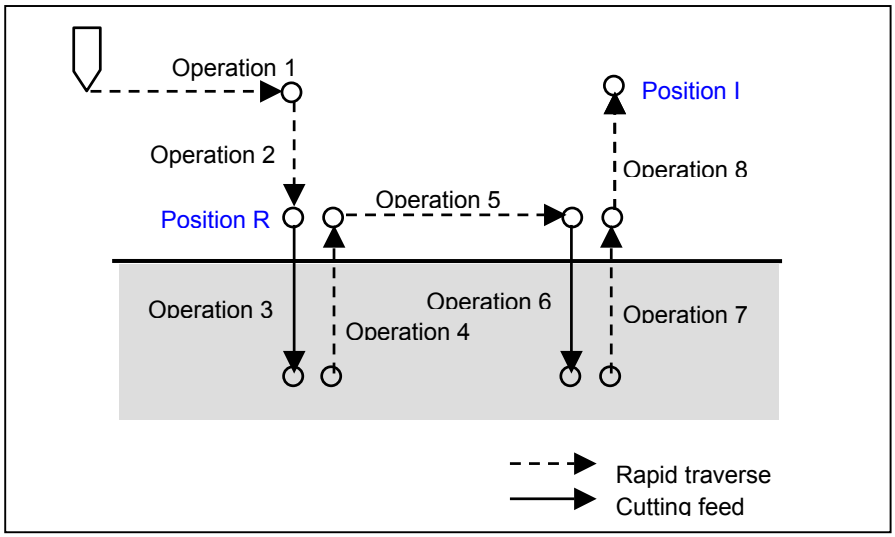


Fig. 4.10.2 (i) Machining operation for [point I - 1]

(2) [point I - 2]:
The tool returns to the point I level before moving from a hole to another hole, including the last operation.
For the first hole position, machining restarts from the block immediately before point I (start point of operation 1). For the second or subsequent hole position, machining restarts from the hole position immediately before point I (start point of operation 6).

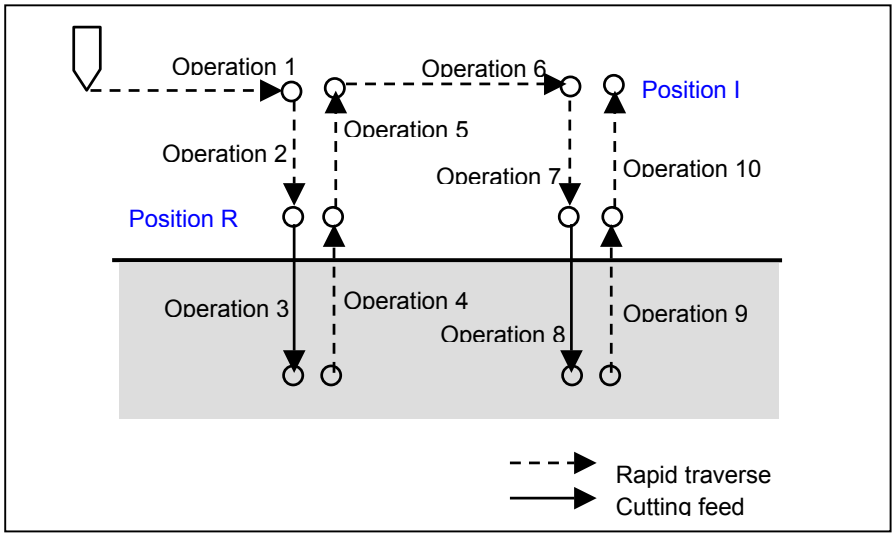


Fig. 4.10.2 (j) Machining operation for [point I - 2]

(3) [point I - 3]:
The tool returns to the point R level before moving from a hole to another hole, including the last operation.
For the first hole position, machining restarts from the block immediately before point R (start point of operation 2). For the second or subsequent hole position, machining restarts from the hole position immediately before point R (start point of operation 5).

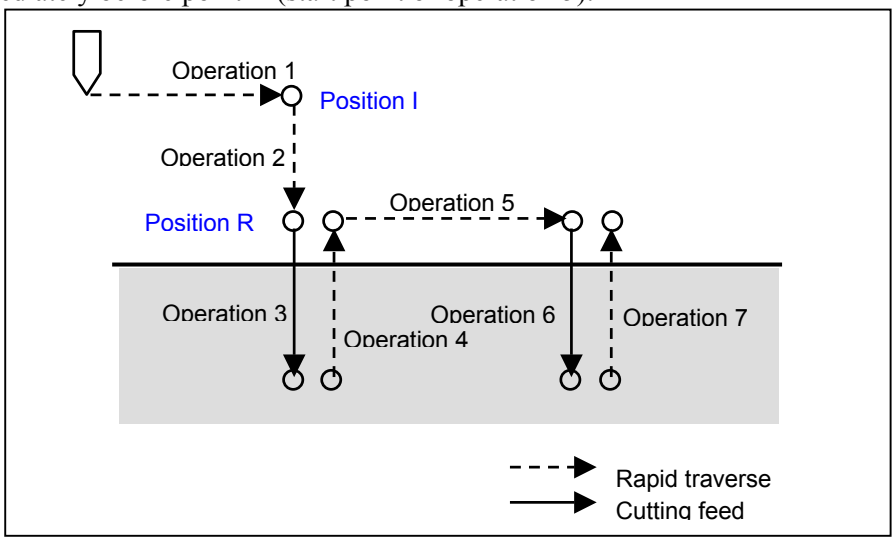


Fig. 4.10.2 (k) Machining operation for [point I - 3]

- Warning

Related warning messages are listed below.

Table 4.10.2 (e) Related warning messages

Message	Description
EXECUTING OPERATION	Soft key [RESTRT] was pressed during automatic operation. After the completion of automatic operation, press soft key [RESTRT].
IT IS NOT EXECUTED	No hole machining cycle has been operated. After performing automatic operation, press soft key [MACHIN CYCLE].

Notes

(1) This function is available only for a hole machining cycle. If an attempt is made to perform quick program restart for a machining cycle for other than hole machining, warning message “IT IS NOT EXECUTED” is displayed.
(2) Machining can be restarted only from an executed block. Hole position information for restart is stored when each block is executed. The memory in which the information is stored is called program restart memory.
(3) Information on up to 50 restart points can be stored in the program restart memory. For a multi-path system, information on up to 50 restart points are stored for each path.
(4) If the program restart memory becomes full, hole position information is deleted from the older one and the information on a new hole position is overwritten.
(5) The program restart memory is kept stored after power-on.
(6) The program restart memory is cleared when automatic operation is performed in the MEM or DNC mode after a reset.
(7) After execution, if the machining program is modified and program restart is performed, the restart position is changed. If the machining program is modified, be sure to operate the program, then perform program restart.
(8) The restart position of a hole machining cycle is the block immediately before point R or I.
(9) For a multi-path system, specify the restart position so that the waiting M codes for each path are consistent.

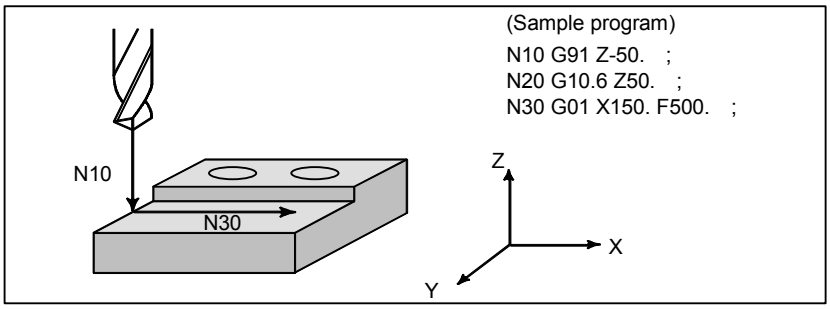
4.11 TOOL RETRACT AND RECOVER

The tool can be retracted from a workpiece to replace the tool, if damaged during machining, or to check the status of machining. Then, the tool can be returned to restart machining efficiently.

Procedure for tool retract and recover

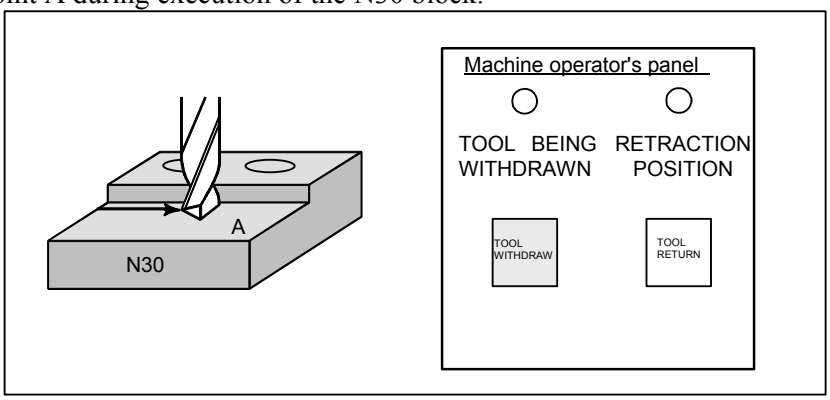
Procedure1 - Programming

Specify a retraction axis and distance in command G10.6IP_ beforehand. In the sample program below, the N20 block specifies that the Z-axis is the retraction axis and the retraction distance is to be 50 mm.

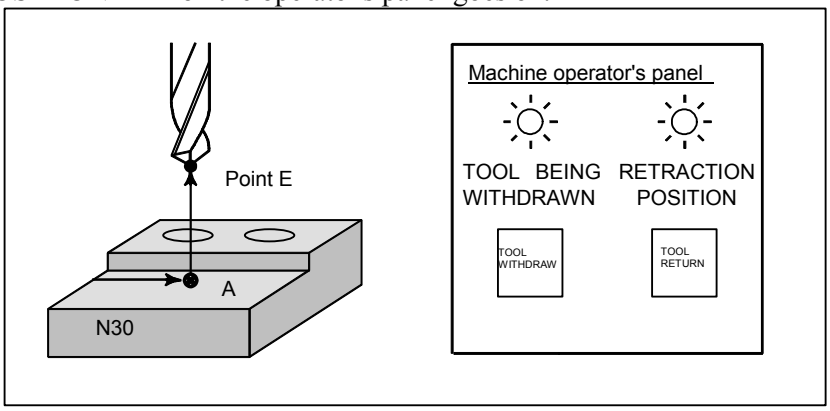


Procedure 2 - Retract

Suppose that the TOOL WITHDRAW switch on the machine operator's panel is turned on when the tool is positioned at point A during execution of the N30 block.



Next, the tool withdrawal mode is set and the TOOL BEING WITHDRAWN LED goes on. At this time, automatic operation is temporarily halted. The tool is then retracted by the programmed distance. If point A is the end point of the block, retraction is performed after automatic operation is stopped. Retraction is based on linear interpolation. The dry run feedrate is used for retraction. Upon completion of retraction, the RETRACT POSITION LED on the operator's panel goes on.



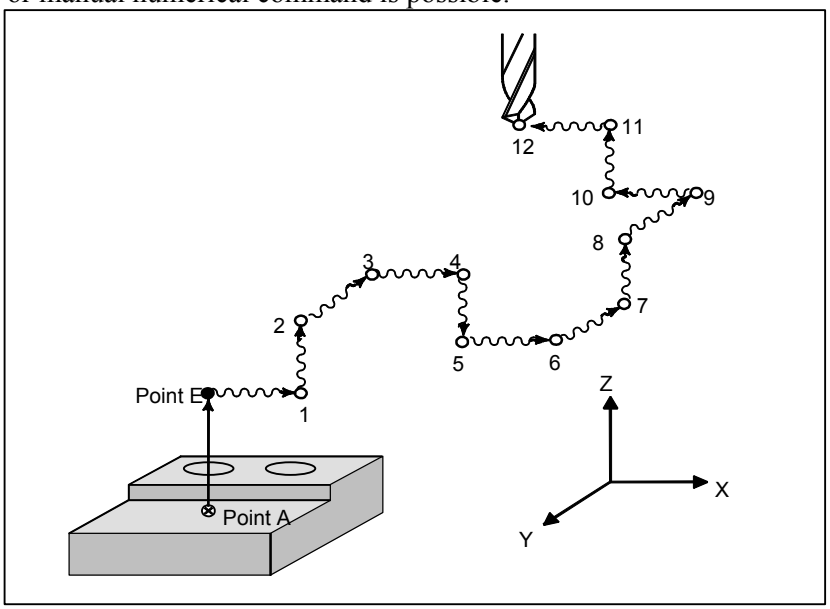
During retraction, the screen displays PTRR and STRT.



- PTRR blinks in the field for indicating states such as the program editing status.
- STRT is displayed in the automatic operation status field.
- MTN is displayed in the field for indicating status such as movement along an axis.

Procedure 3 - Withdrawal

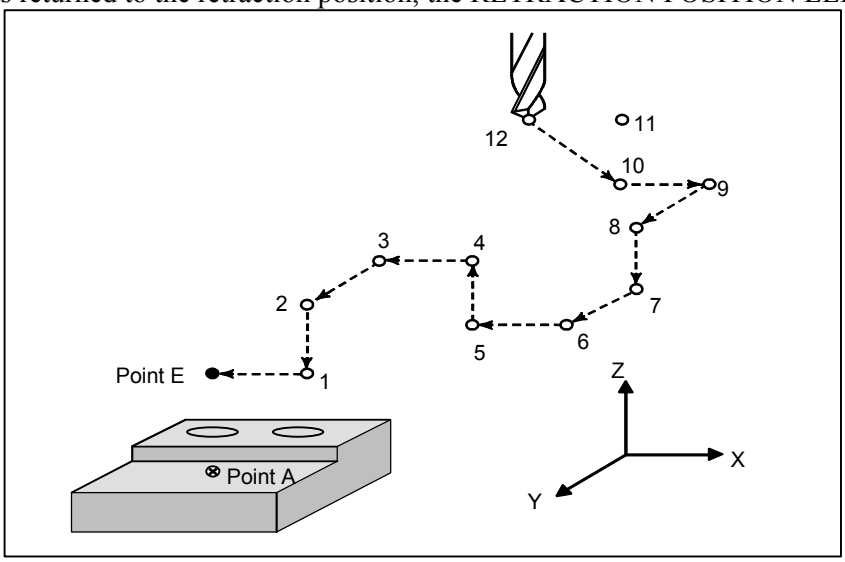
Set the manual operation mode, then withdraw the tool. For manual operation, either jog feed incremental feed, handle feed, or manual numerical command is possible.



Procedure 4 - Return

After withdrawing the tool and any additional operation such as replacing the tool, move the tool back to the previous retraction position. To return the tool to the retraction position, return the mode to automatic operation mode, then turn the TOOL RETURN switch on the operator's panel on then off again. The tool returns to the retraction position at the dry run feedrate, regardless of whether the dry run switch is on or off.

When the tool has returned to the retraction position, the RETRACTION POSITION LED comes on.



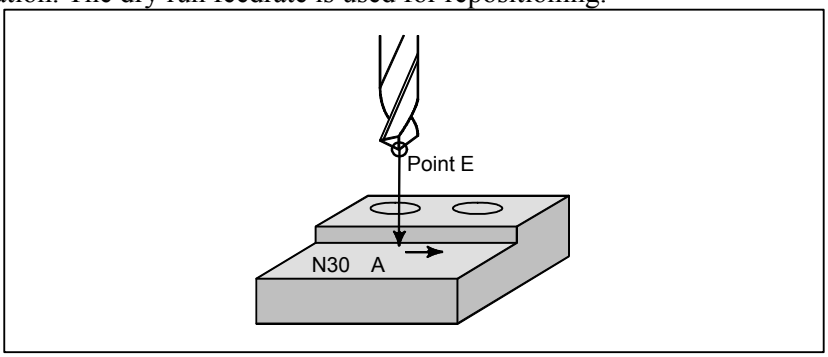
During return operation, the screen displays PTRR and MSTR.



- PTRR blinks in the field for indicating states such as program editing status.
- MSTR is displayed in the automatic operation status field.
- MTN is displayed in the field for indicating states such as movement along an axis.

Procedure 5 - Repositioning

While the tool is at the retraction position (point E in the figure below) and the RETRACTION POSITION LED is on, press the cycle start switch. The tool is then repositioned at the point where retraction was started (i.e. where the TOOL WITHDRAW switch was turned on). Repositioning is based on linear interpolation. The dry run feedrate is used for repositioning.



Upon completion of repositioning, the tool withdraw mode is cancelled, and the TOOL BEING WITHDRAWN LED goes off and restart N30.

Limitation

1 If the origin, presetting, workpiece origin offset value (or external workpiece origin offset value), or workpiece coordinate shift amount (for a lathe system) is changed after the retraction position is specified with G10.6 in the absolute mode, the change is not reflected in the retraction position. After such changes are made or the workpiece origin offset value (or external workpiece origin offset value) or workpiece coordinate shift amount (for a lathe system) is changed, respectively the retraction position with G10.6.

2 When retracting the tool manually in the tool withdrawal mode, do not use the machine lock, mirror-image, or scaling function.

3 Tool retract and recover cannot be performed along axes that are in the modes below.
- Superimposed control

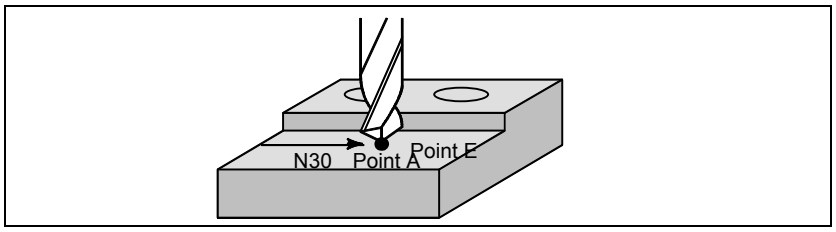
> ⚠ **WARNING**
> The retraction axis and retraction distance specified in G10.6 must be changed in an appropriate block according to the figure being machined. Be very careful when specifying the retraction distance; an incorrect retraction distance may damage the workpiece, machine, or tool.

4.11.1 Retract

Explanation

- When no retraction distance is specified

If no retraction distance or direction required for retraction are specified, retraction is not performed when the TOOL WITHDRAW switch on the operator's panel is turned on. Instead, the block being executed in automatic operation is interrupted (automatic operation is held or stopped). In this state, the tool can be withdrawn and returned.



- Retraction from the automatic operation hold or stop state

When the single block switch is turned on during automatic operation, or the TOOL WITHDRAW switch is turned on after the automatic operation hold or stop state is set by feed hold: Retraction is performed, then the automatic operation hold or stop state is set again.

- Stopping retraction

During retraction, feed hold operation is ignored. However, reset operation is enabled (retraction is stopped at reset). When an alarm is issued during retraction, the retraction is stopped immediately.

- Repositioning immediately after retraction

After retraction is completed, tool repositioning can be started without performing the withdraw and return operations.

4.11.2 Withdrawal

Explanation

- Axis selection

To move the tool along an axis, select the corresponding axis selection signal. Never specify axis selection signals for two or more axes at a time.

- Path memorization

When the tool is moved in manual operation along an axis, the control unit memorizes up to ten paths of movements. If the tool is stopped after being moved along a selected axis and is then moved along another selected axis, the position where this switch takes place is memorized. After ten paths have been memorized, the control unit does not memorize any additional switching points.

- Reset

Upon reset, memorized position data is lost and the tool withdraw mode is cancelled.

NOTE

If an attempt is made to move the tool simultaneously along two axes using the manual numeric command in the tool withdrawal mode, an alarm PS0015, “TOO MANY SIMULTANEOUS AXES” is issued.

4.11.3 Return

Explanation

- Return path

When there are more than ten return paths, the tool first moves to the tenth position, then to the ninth position, then to the eighth position, and so forth until the retraction position is reached.

- Single block

The single block switch is enabled during return operation. If the single block switch is turned off, continuous return operation is performed. If the single block switch is turned off, the tool stops at each memorized position. In this case, return operation can be resumed by turning the TOOL RETURN switch on then off again.

- Interruption of return operation

When an alarm is issued during return operation, return operation stops.

- Feed hold

The feed hold function is enabled during return operation.

4.11.4 Repositioning

Explanation

- Feed hold

The feed hold function is disabled during repositioning.

- Operation after completion of repositioning

The operation after completion of repositioning depends on the automatic operation state present when the TOOL WITHDRAW switch is turned on.

1 When automatic operation is being started
After completion of repositioning, the interrupted execution of the block is resumed.
2 When automatic operation is held or stopped
After completion of repositioning, the tool stops once at the repositioned point, then the original automatic operation hold or stop state is set. When the cycle start switch is pressed, automatic operation is resumed.

4.11.5 Tool Retract and Recover for Threading

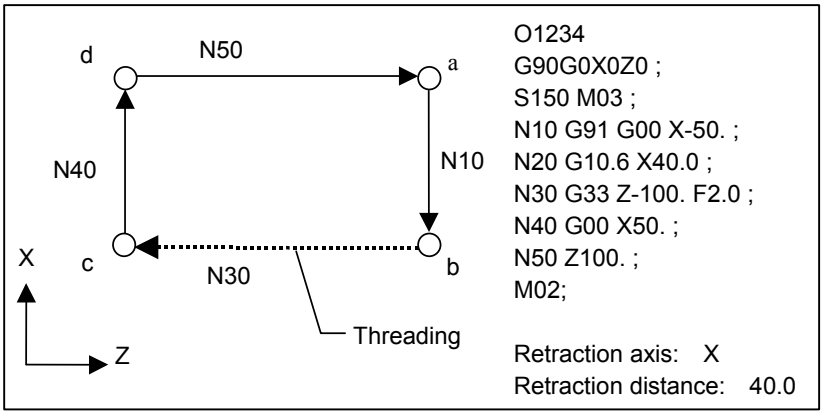
Explanation

- Differences between ordinary tool retract and recover and tool retract and recover for threading

1 During retraction, chamfering is performed between the specified retraction axis and threading axis.
2 After retraction, one block that does not specify threading is executed and the tool stops.
3 When the major axis for threading is specified as the retraction axis, retraction is not performed by turning the TOOL WITHDRAW switch on. In this case, after a block that does not specify threading is executed, an alarm PS0429, “ILLEGAL COMMAND IN G10.6” is issued and the tool stops.
4 As repositioning, the tool is returned to the position specified in the first block that does not specify threading.

- Operation procedure

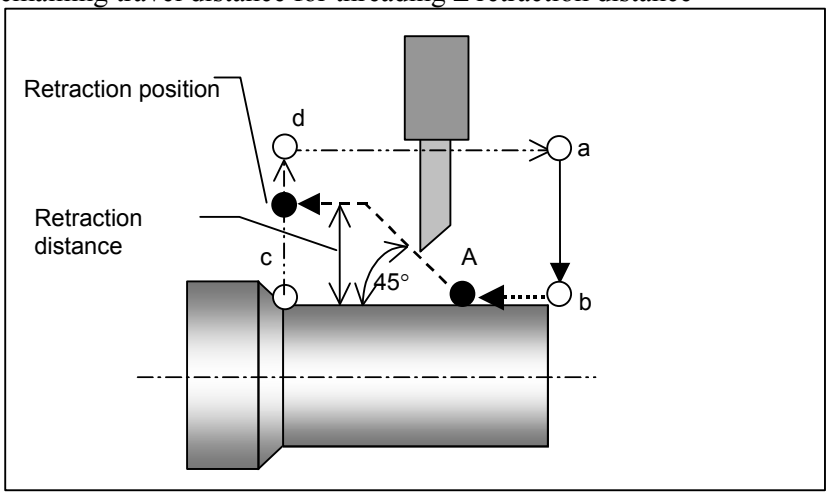
1 Specify a retraction axis and retraction distance in command “G10.6IP- -;”.



2 Turn the TOOL WITHDRAW switch during the execution of a threading command block.

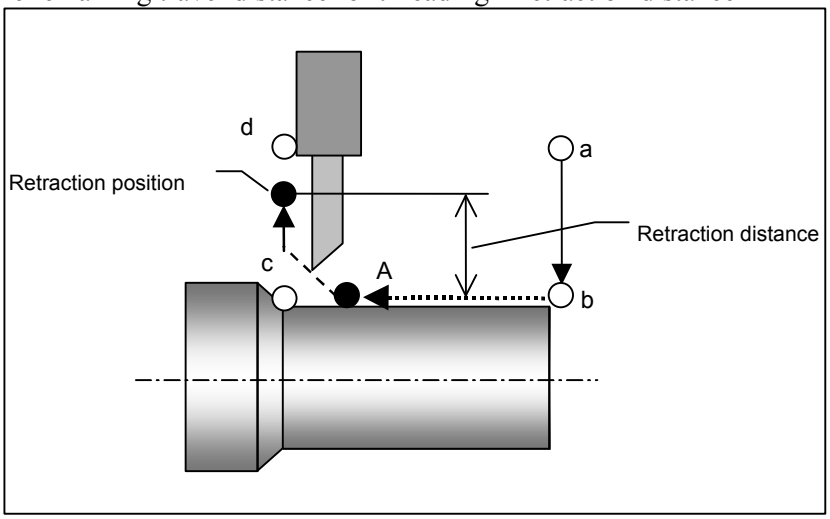
3 The tool withdrawal mode is set and retraction is performed. Chamfering at 45 degrees is performed between the retraction axis and major axis for threading using the retraction distance as the chamfering amount during retraction.
Details of retraction differ depending on whether the remaining travel distance for the threading command is smaller than the retraction distance when the TOOL WITHDRAW switch is turned on as follows:

(1) When remaining travel distance for threading ≥ retraction distance



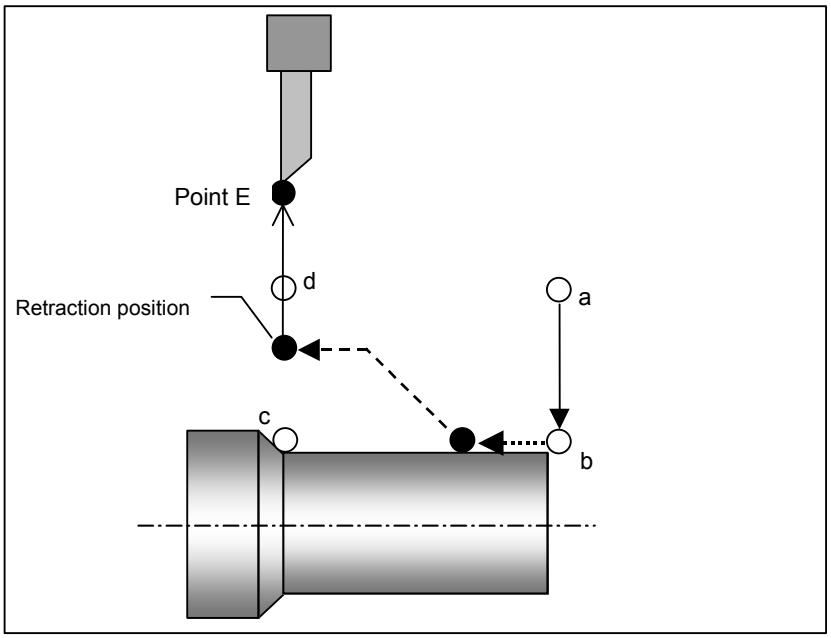
When the position where 45-degree chamfering by the retraction distance ends does not exceed the threading end position (c), the tool moves to the threading end position after the termination of chamfering.

(2) When the remaining travel distance for threading < retraction distance



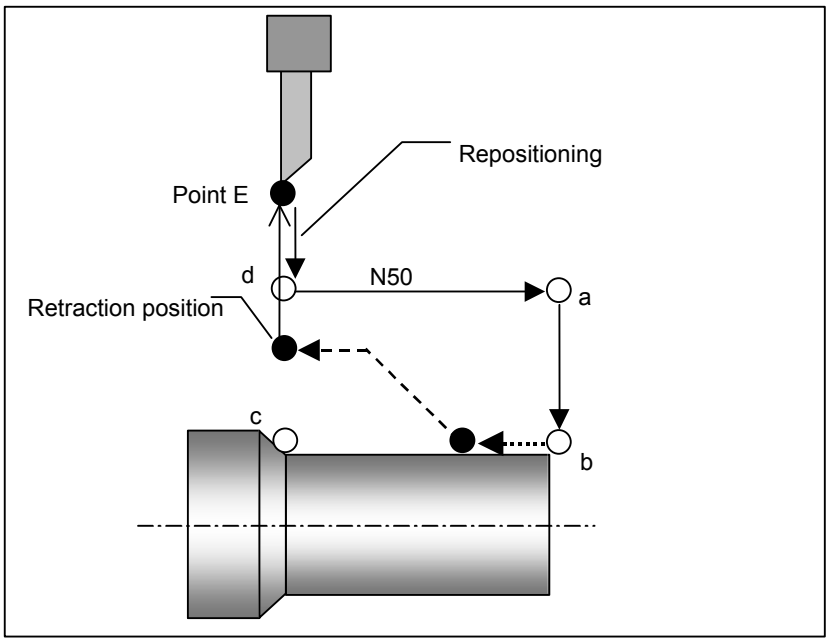
When the position where 45-degree chamfering by the retraction distance ends exceeds the threading end position (c), the tool moves to the retraction position along the retraction axis after it reaches the threading end position.

4 After retraction is completed, the next block that does not specify threading is executed and the tool stops.



In this example, “X50.0” is specified in the first block that does not specify threading in the incremental mode, the tool moves to point E and stops.
If the major axis for threading is specified as the retraction axis, the block that does not specify threading is executed without performing retraction, an alarm PS0429 is issued, and the tool stops.

5 As repositioning, the tool returns to the position specified in the first block that does not specify threading.



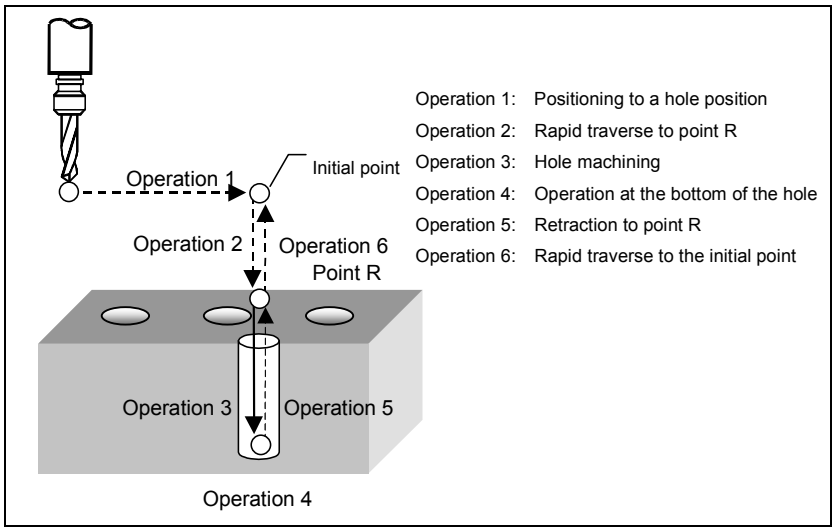
In this example, the repositioning position is point d. Automatic operation after repositioning starts at the N50 block.

4.11.6 Operation Procedure for a Canned Cycle for Drilling

Explanation

- **Retract**

When the TOOL WITHDRAW switch is turned on during a canned cycle for drilling (abbreviated as a canned cycle below), retraction is performed depending on the cycle operation being executed at that time.



1 During operation 1, the tool moves by the retraction distance specified in G10.6 in the same way as for ordinary retraction.
2 During operation 2, the tool stops operation 2, moves to the initial point, and stops.
3 During operation 3, the tool stops operation 3, executes cycle operations 4, 5, and 6 from that position, and stops at the initial point.

4 During operation 4, 5, or 6, the tool continues the operation and stops at the initial point.
When the TOOL WITHDRAW switch is turned on during operation 2 to 6, the tool does not move according to the retraction specified in G10.6. After the TOOL WITHDRAW switch is turned on and the tool moves to the initial point, however, the tool withdrawal mode is set.
When the second or subsequent canned cycle is being executed and the TOOL WITHDRAW switch is turned on during operation 2 to 6, the retraction position differs depending on G98 (return to initial level) or G99 (return to point R level).
- G98 (return to initial level): The tool moves to the initial level.
- G99 (return to point R level):The tool moves to the point R level.

5 During operation 2 to 6, the tool also moves to the initial point and stops when the TOOL WITHDRAW switch is turned on without the G10.6 command specified.

- Repositioning

When the tool is at the retraction position and the cycle start switch is pressed, repositioning is performed for the canned cycle.

1 Repositioning performed when the TOOL WITHDRAW switch is turned on during operation 1
After the completion of repositioning, automatic operation is resumed in the same way as for ordinary repositioning.

2 Repositioning performed when the TOOL WITHDRAW switch is turned on during operation 2
The canned cycle is reexecuted from operation 2.

3 Repositioning performed when the TOOL WITHDRAW switch is turned on during operation 3
The canned cycle is reexecuted from operation 2.

4 Repositioning performed when the TOOL WITHDRAW switch is turned on during operation 4, 5, or 6
The canned cycle is reexecuted for the same hole position from operation 2.

4.12 MANUAL INTERVENTION AND RETURN

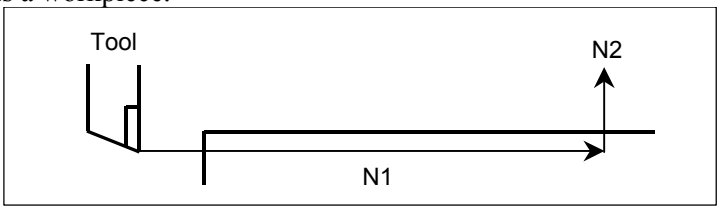
Overview

If you use feed hold to stop the tool from moving an axis during automatic operation and restarts the tool after manual intervention, for example, for checking a cutting surface, the tool can resume automatic operation after automatically returning to the pre-intervention position.
This function is optional.

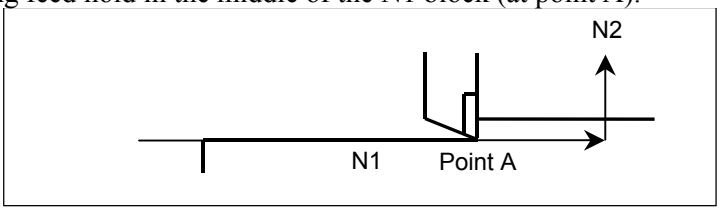
Explanation

Setting bit 0 (MIT) of parameter No. 7001 enables manual intervention and return.
The sequence of manual intervention and return is as Fig. 4.12 (a):

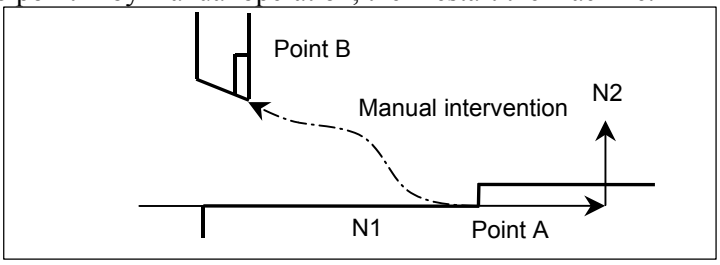
1. The N1 block cuts a workpiece.



2. Stop the tool using feed hold in the middle of the N1 block (at point A).



3. Retract the tool to point B by manual operation, then restart the machine.



4. After the tool automatically returns to point A by non-linear interpolation type positioning at the dry run feedrate, the remaining move command of the N1 block is executed.

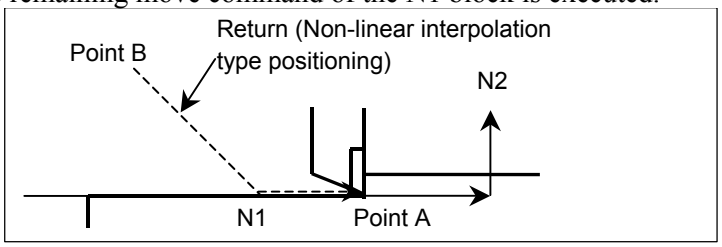
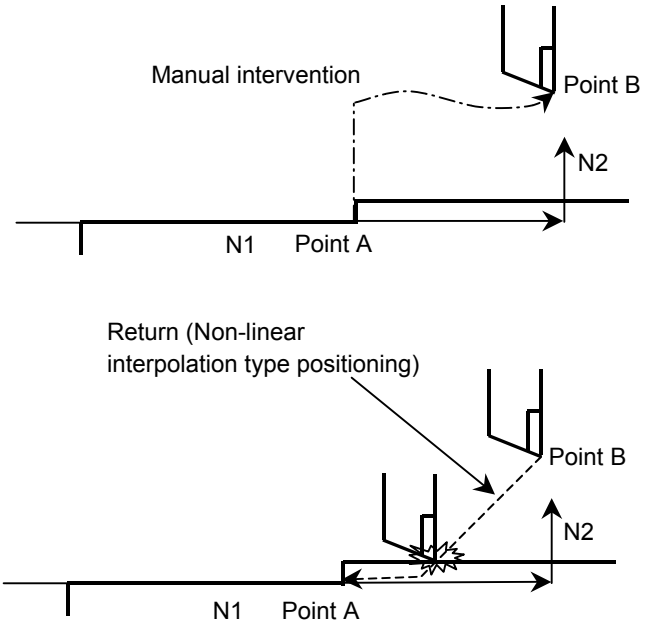


Fig. 4.12 (a) Sequence of manual intervention and return

⚠ WARNING

Manual intervention must be performed correctly with meticulous care, following the machining direction and the shape of the workpiece, not to damage the workpiece, machine, and/or tool.



- Manual absolute on/off

In manual absolute off mode, the tool does not return to the stop position. Instead, the tool moves according to the manual absolute on/off function.

- Return feedrate

For the return operation, the dry run feedrate is used, and the jog feedrate override function is enabled. When manual rapid traverse rate selection signal RT is "1", the return feedrate is not dry run feedrate but rapid traverse rate.

- Return operation

Return operation is performed according to non-linear interpolation type positioning.

- Single block

If the single block stop switch is on during return operation, the tool stops at the stop position once and restarts movement when cycle start is executed again.

- Cancellation

If a reset, alarm, or emergency stop occurs during manual intervention or return operation, the manual intervention and return function is cancelled.

- MDI mode

The manual intervention and return function can be used in the MDI mode as well.

- Relation to other functions

In other functions such as canned cycle, the manual intervention and return function is effective.

- PMC axis control

If intervention is performed under PMC axis control after feed-holding of automatic operation, return operation does not apply to the axis moving under PMC axis control when the return operation starts.

If the return operation is started with the axis under PMC axis control being stopped after having completed the PMC axis control command, however, the return operation is performed by the amount of movement by PMC axis control.
When PMC axis control is used for an axis unrelated to the program command, set bit 0 (NRT) of parameter No. 10410 to 1 to exclude the axis completely from return operation by the manual intervention and return function.

Limitations

- Enabling/disabling manual intervention and return

The manual intervention and return function is enabled only when manual intervention is performed with the automatic operation hold lamp on. If feed hold and manual intervention are performed with no travel distance remaining, this function has no effect and the operation is performed according to the manual absolute on/off function.

- Offset

When the tool is replaced using manual intervention for a reason such as tool damage, even if the amount of offset is changed when restarting in the middle of the interrupted block, the change is not reflected in machining.

- Machine lock, mirror image, and scaling

When performing manual intervention and return, never use the machine lock, mirror image, or scaling function.

- 3-dimensional coordinate system conversion, tilted working plane command, and tool center point control for 5-axis machining

Alarm PS5219, “CAN NOT RETURN” is issued when manual intervention and the return is attempted during execution of 3-dimensional coordinate system conversion, tilted working plane command, or tool center point control.

- Parking

Do not switch the parking signal during manual intervention and return (for example, after feed hold stop). The tool may not return in the right position when the parking signal is switched during manual intervention and return.

- Manual intervention and return by multi-path system

If cycle start is executed at the same time on two or more paths in a multi-path system, the path that performed manual intervention does return operation. However, the path that did not perform manual intervention restarts processing without doing return operation.

- Twin table

Manual intervention and return cannot be performed under twin table control.

- Axis for which return operation is not performed

When the axis is in the following conditions, return operation of the axis is not performed.

- Spindle positioning
- Tapping
- Slave axis of axis synchronous control
- Slave axis of synchronous control
- Spindle control by servo motor
- Spindle control by Cs contour control
- Polygon axis
- Chopping axis

NOTE
Under synchronous control, manual intervention and return can be performed for the slave axis only when bit 2 (PKUx) of parameter No. 8162 is 1 and the master axis is parking.

4.13 RETRACE

M

Overview

The tool can retrace the path along which the tool has moved so far (reverse execution). Furthermore, the tool can move along the retraced path in the forward direction (forward reexecution). After forward reexecution is performed until the tool reaches the position at which reverse execution started, machining is continued as programmed.

Procedure

- Forward execution → reverse execution

To perform forward execution of a program, set the "REVERSE" switch on the machine operator's panel to off, then perform a cycle start operation. If the "REVERSE" switch on the machine operator's panel is set to on, reverse execution or the end of reverse execution results.
To perform reverse execution of a program, use one of the following three methods:

1) Set the "REVERSE" switch on the machine operator's panel to on during forward execution of a block.
2) Perform a single block stop operation during forward execution, then set the "REVERSE" switch on the machine operator's panel to on.
3) Perform a feed hold stop operation during forward execution, then set the "REVERSE" switch on the machine operator's panel to on.

When method 1) is used, reverse execution starts after the end of the block being executed (after execution up to the single block stop position). Reverse execution does not start as soon as the "REVERSE" switch on the machine operator's panel is set to on.

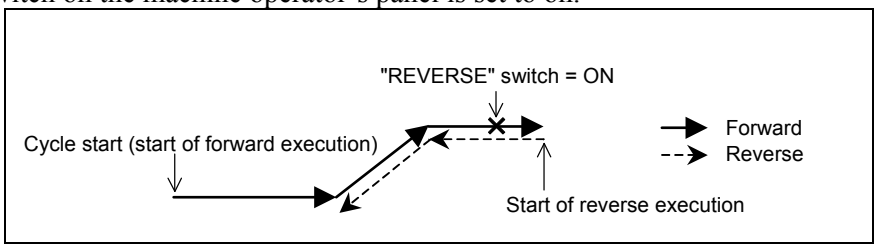


Fig. 4.13 (a)

If functions for gas cutting machine are enabled, however, only when linear interpolation (G01), circular interpolation (G02, G03), or skip function (G31) is specified, reverse execution starts at the position at which the "REVERSE" switch is set to on.

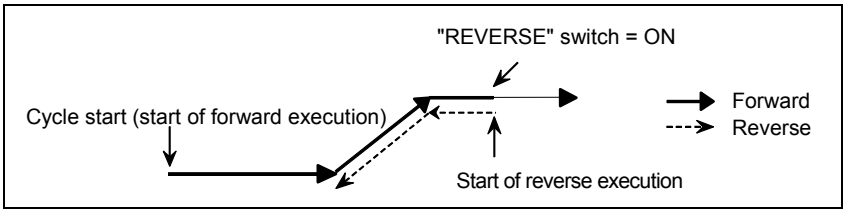


Fig. 4.13 (b)

When method 2) is used, performing a cycle start operation starts reverse execution from the position at which a single block stop takes place.

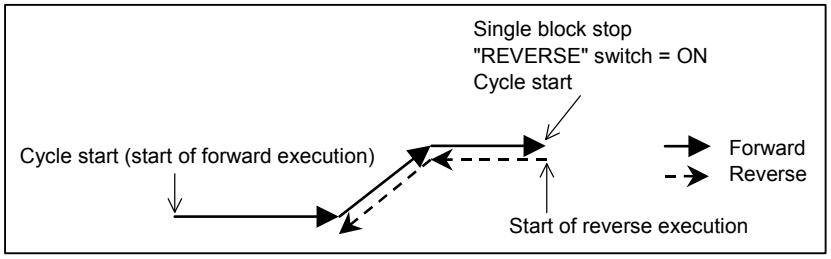


Fig. 4.13 (c)

When method 3) is used, performing a cycle start operation starts reverse execution from the position at which a feed hold stop takes place.

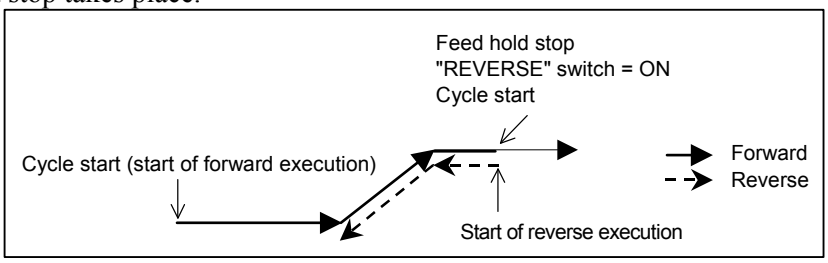


Fig. 4.13 (d)

- Reverse execution → forward reexecution

To perform forward reexecution of a program, use one of the following three methods:

1) Set the “REVERSE” switch on the machine operator’s panel to off during reverse execution of a block.
2) Set the “REVERSE” switch on the machine operator’s panel to off after a single block stop takes place during reverse execution.
3) Set the “REVERSE” switch on the machine operator’s panel to off after a feed hold stop takes place during reverse execution.

When method 1) is used, forward reexecution starts after the block being executed ends (after execution up to the position at which a single block stop takes place). Forward reexecution does not start as soon as the “REVERSE” switch on the machine operator’s panel is set to off.

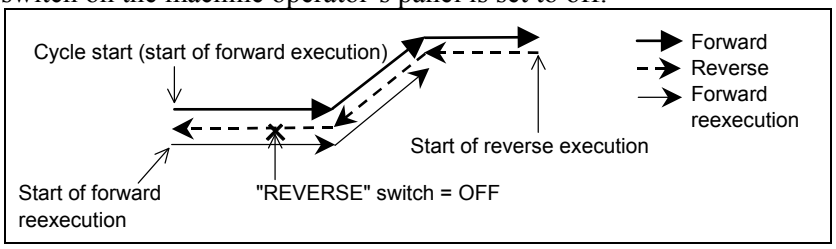


Fig. 4.13 (e)

If functions for gas cutting machine are enabled, however, only when linear interpolation (G01), circular interpolation (G02, G03), or skip function (G31) is specified, forward reexecution starts at the position at which the “REVERSE” switch is set to 0.

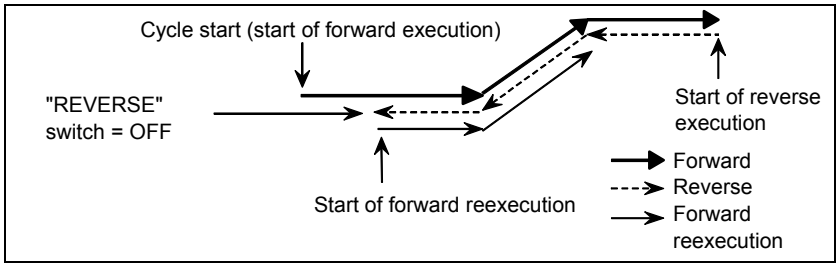


Fig. 4.13 (f)

When method 2) is used, performing a cycle start operation starts forward reexecution from the position at which a single block stop takes place.

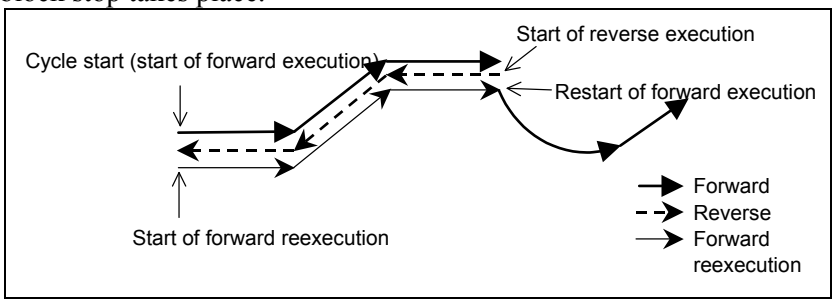


Fig. 4.13 (g)

When method 3) is used, performing a cycle start operation starts forward reexecution from the position at which a feed hold stop takes place.

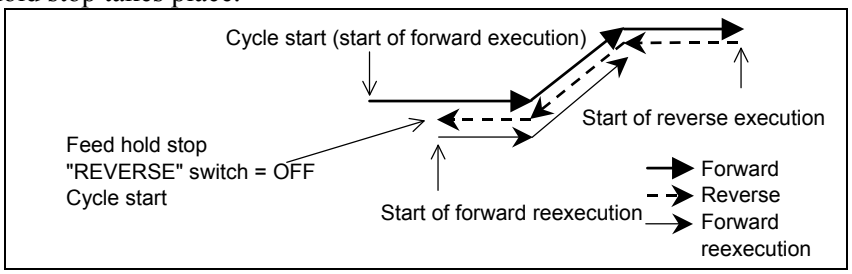


Fig. 4.13 (h)

- Reverse execution → end of reverse execution → forward reexecution

When a block to be executed is no longer present during reverse execution (when reverse execution has been performed up to the block where forward execution started, or when forward execution has not yet been performed), the reverse execution end state is entered, and operation stops.
Even when a cycle start operation is performed while the “REVERSE” switch on the machine operator’s panel is held on, operation is not performed, and the reverse execution end state is maintained. Forward reexecution (or forward execution) is started by setting the “REVERSE” switch on the machine operator’s panel to off then performing a cycle start operation.

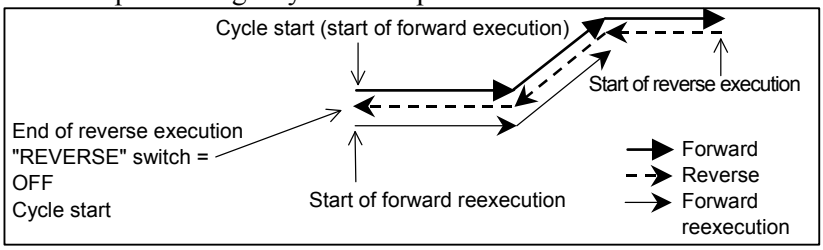


Fig. 4.13 (i)

- Forward reexecution → forward execution

After forward reexecution is performed up to the block at which reverse execution started, forward execution starts automatically, and commands are read from the program again and executed. No particular operation is required.

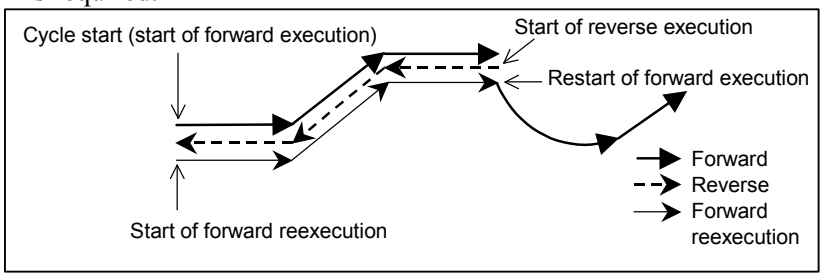


Fig. 4.13 (j)

If reverse execution was performed after feed hold stop, forward reexecution ends when the feed hold stop position is reached, then forward execution is performed. Also if single block operation was performed, forward reexecution ends at the single block stop position.

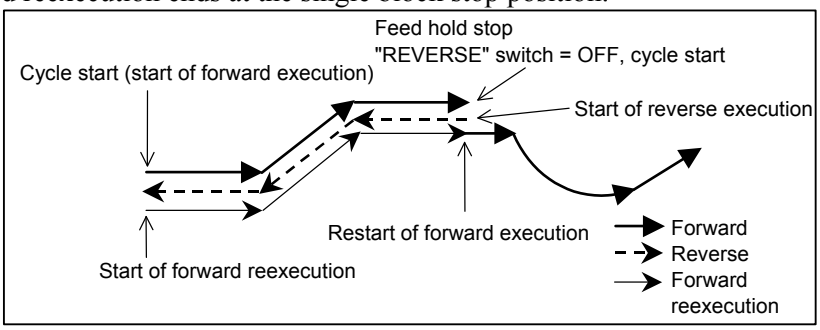


Fig. 4.13 (k)

Explanation

- Reverse execution and forward execution

Usually in automatic operation, a program is executed in the programmed order. This is called forward execution. This function allows a program executed by forward execution to be executed in the reverse direction. This is called reverse execution. Reverse execution allows the tool to retrace the path along which the tool has moved during forward execution.
Reverse execution of a program can be performed only for blocks that have been executed by forward execution.
Furthermore, in single block operation, reverse execution can also be performed on a block-by-block basis .

- Forward reexecution

Blocks that have been executed by reverse execution can be reexecuted in the forward direction up to the block from which reverse execution started. This is called forward reexecution. Forward reexecution allows the tool to retrace the same tool path as in forward execution until the position at which reverse execution started is reached.
After the block from which reverse execution started is reached, the program is executed again in the programmed order (forward execution).
Furthermore, in single block operation, forward reexecution can also be performed on a block-by-block basis.

- End of reverse execution

When a block to be executed is no longer present during reverse execution (when stored blocks have all been executed during reverse execution, or when forward execution has not yet been performed), operation stops. This is called the end of reverse execution.

- Status indication

During reverse execution, characters "RVRS" blink on the screen. During forward reexecution, characters "RTRY" blink to indicate that forward reexecution is in progress. The "RTRY" indication is kept blinking until the block at which reverse execution started is reached and normal operation starts (until forward execution is restarted).
When a block to be executed is no longer present during reverse execution, or if an attempt is made to perform reverse execution for a block that cannot be executed by reverse execution, characters "RVED" blink, notifying the user that reverse execution can no longer be performed.

- Number of blocks that can be executed by reverse execution

Up to about 100 blocks can be executed by reverse execution. Depending on the specified program, the maximum number of executable blocks may decrease.

- Reset

A reset operation (the RESET key on the MDI unit, the external reset signal, or the reset & rewind signal) clears the blocks stored for reverse execution.

- Feedrate

A feedrate to be applied during reverse execution can be specified in parameter No. 1414. If this parameter is set to 0, the feedrate in reverse execution is assumed to be the same as that in forward execution. Rapid traverse, however, is performed always at the rapid traverse rate, regardless of the setting of this parameter.
The feedrate in forward reexecution is always the same as that in forward execution.
In reverse execution or forward reexecution, feedrate override, rapid traverse override, and dry run are allowed.

- Start of reverse execution or forward reexecution after the end of a block

In a block for rapid traverse (G00), linear interpolation (G01), circular interpolation (G02, G03), dwelling (G04), skip cutting (G31), or an auxiliary function in an automatic operation mode (memory operation, part program operation, or MDI operation), reverse execution or forward reexecution can be started. However, reverse execution and forward reexecution do not start as soon as the reverse execution signal status is changed. When the block has ended, that is, after a movement, dwelling, or an auxiliary function is completed, reverse execution or forward reexecution starts.

- Start of reverse execution or forward reexecution after feed hold stop

When a feed hold stop operation is performed during execution of rapid traverse (G00), linear interpolation (G01), circular interpolation (G02, G03), or skip cutting (G31), then the reverse execution signal status is changed and operation is restarted, reverse execution or forward reexecution can be started immediately from the stop position. This cannot be performed when dwelling (G04) or an auxiliary function is being executed.

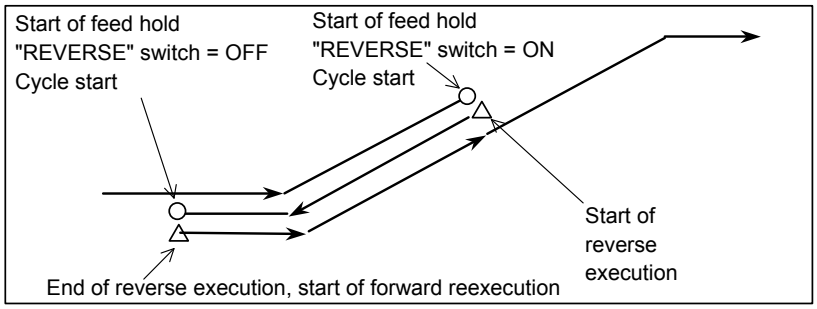


Fig. 4.13 (l)

When reverse execution is started after feed hold stop operation, the part from the start point of that block to the feed hold stop position is stored as one block. Therefore, when forward reexecution is performed with the single block switch set to 1, a single block stop takes place as soon as the position at which reverse execution started is reached.

- Start of reverse execution or forward reexecution after single block stop

After a single block stop takes place, reverse execution or forward reexecution can be started immediately when the reverse execution signal status is changed and restart operation is performed.

Limitation

- Blocks that cannot be executed by reverse execution

In the modes listed below, reverse execution cannot be performed.
When one of these commands appears during reverse execution, reverse execution ends immediately and "RVED" is displayed.

- Involutes interpolation (G02.2, G03.2)
- Exponential interpolation (G02.3, G03.3)

- 3-dimensional circular interpolation (G02.4, G03.4)
- NURBS interpolation (G06.2)
- Cylindrical interpolation (G07.1,G107)
- Polar coordinate interpolation (G12.1, G13.1,G112,G113)
- Polar coordinate command (G16)
- Functions related thread cutting (G33,G34,G35,G36)
- 3-dimensional cutter compensation (G41.2, G41.3, G42.2, G42.3)
- Tool length compensation in tool axis direction (G43.1)
- Tool center point control (G43.4, G43.5)
- Polygon turning (G50.2, G51.2)
- Tool axis direction control (G53.1)
- Single direction positioning (G60)
- Tapping mode (G63)
- Tapping cycle (G84,G74)
- Rigid tapping cycle (G84,G74,G84.2,G84.3)
- Fine boring cycle (G76)
- Back boring cycle (G87)

It is impossible to perform reverse execution for blocks specifying the commands listed below. If one of these commands appears during reverse execution, reverse execution ends immediately and "RVED" is displayed.
Some of these commands turn a mode on and off. It is possible to start reverse execution and perform forward reexecution in a mode set by such a command. However, if a block that turns the mode on or off is reached during reverse execution, the reverse execution ends at that block, and "RVED" is displayed.

- Functions related AI contour control (G05,G05.1,G08)
- HRV3 on/off (G05.4)
- Hypothetical axis interpolation (G07)
- Inch/metric conversion (G20, G21)
- Stored stroke check on/off (G22, G23)
- Functions related reference position return (G27, G28, G29, G30)
- Floating reference position return (G30.1)
- 3-dimensional coordinate system conversion (G68, G69)
- Feature coordinate system (G68.2)
- Figure copying (G72.1,G72.2)
- Chopping (G81.1)
- Index table indexing
- Cs contouring control
- Spindle positioning

- Manual intervention

To execute a program in the reverse direction after a feed hold stop or single block stop, when manual intervention is performed after the stop, make a return to the original position and then turn on the reverse signal. Movement by manual intervention is ignored during reverse execution and forward reexecution.
If manual intervention is performed during reverse execution or forward reexecution, the amount of manual intervention is added to the coordinate system at a restart after a stop due to a feed hold or single block during forward execution after the end of forward reexecution. Whether to add the amount of manual intervention follows the manual absolute switch.

- Single block stop position

A block that is internally generated by the control unit is also treated as one block during reverse execution.

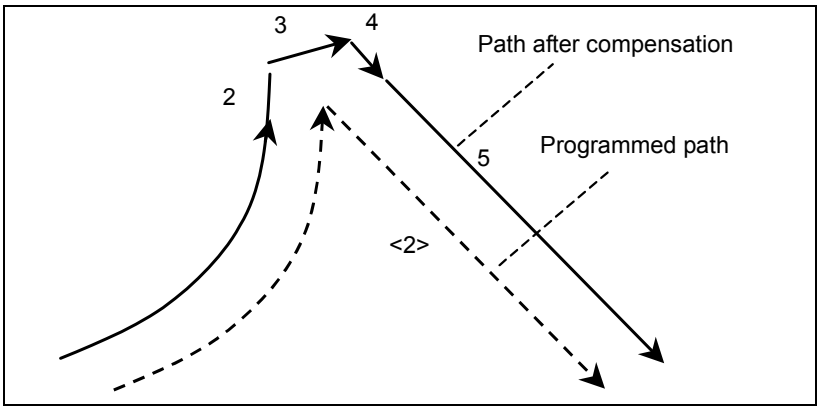


Fig. 4.13 (m) Path when cutter compensation is applied

In the above example, the program specifies two blocks, but in actual operation, move commands for five blocks are generated.
In such a case, positions at which a single block stop takes place may differ between forward execution and reverse execution.

- Positioning (G00)

When non-linear type positioning is performed (bit 1 (LRP) of parameter No. 1401 is set to 0), the tool path in reverse execution and that in forward execution do not match. The tool path in forward reexecution is the same as that in forward execution.
When linear type positioning is performed (bit 1 (LRP) of parameter No. 1401 is set to 1), the tool path in reverse execution is the same as that in forward execution.

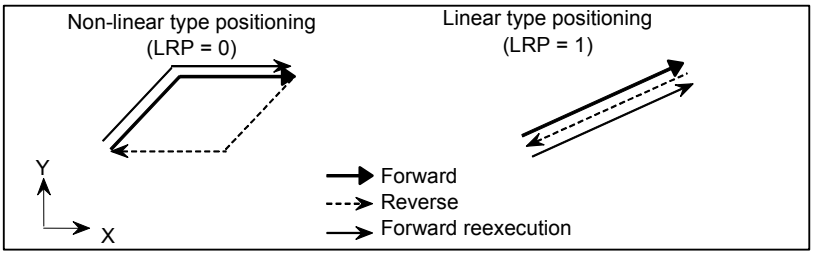


Fig. 4.13 (n)

- Dwell command (G04)

During reverse execution or forward reexecution, the dwell command (G04) is executed in the same way as in normal operation.

- Programmable data input (G10)

Tool compensation values, parameters, pitch error data, workpiece origin offsets, and tool life management values set or modified by programmable data input (G10) are ignored during reverse execution and forward reexecution.

- Skip function (G31) and automatic tool length compensation (G37)

The skip signal and the automatic tool length measurement signal are ignored during reverse execution and forward reexecution. During reverse execution and forward reexecution, the tool moves along the path that the tool has actually passed during forward execution.

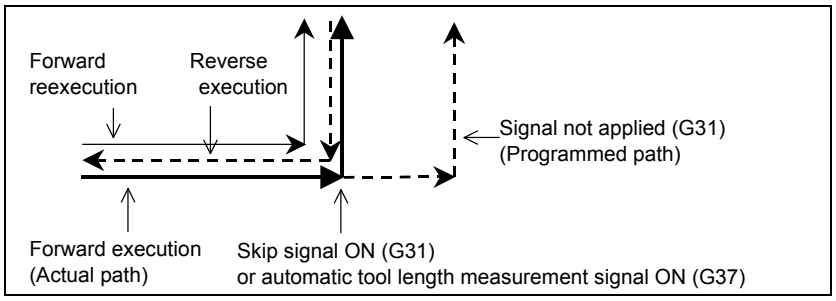


Fig. 4.13 (o)

- Setup of a coordinate system (G92, G54 to G59, G54.1P_, G52, and G92.1)

When setup of a coordinate system (G92, G54 to G59, G54.1P_, and G52) is specified during reverse execution, the indicated current position may differ from the position indicated during forward execution. However, the actual machine position does not differ.

- Mirror image

When a block to which a mirror image is applied by programmable mirror image (G50.1, G51.1) is executed during reverse execution, the tool moves along the actual path resulting from the application of a mirror image in the reverse direction.
When a mirror image is applied to a block by setting or a machine signal, the block with the mirror image not applied is stored. Mirror image application by setting or a machine signal is enabled also during reverse execution and forward reexecution. Therefore, during reverse execution and forward reexecution, the mirror image by setting data or machine signal must be turned on and off so that this on/off status and the on/off status during forward execution match.

- Changing offsets

Even when cutter compensation data or tool length offsets are changed during reverse execution or forward reexecution, the change in compensation or offset data does not become valid until forward reexecution ends and normal operation starts. Until then, the tool moves with the offset data that was applied when the block was executed for the first time during forward execution.

- Feedrate clamp

During reverse execution or forward reexecution, feedrate clamp is not performed with parameter No. 1420 (rapid traverse rate) or parameters Nos. 1430 and 1432 (maximum cutting feedrate). It is executed with parameter No. 1414 or at the feedrate assumed during forward execution.
If, for example, the parameters above are set to smaller values during reverse execution or forward reexecution, clamp is not performed with these values, but with parameter No. 1414 or at the feedrate assumed during forward execution.
For clamp at the feedrate assumed during backward execution or forward reexecution, change the feedrate with the external deceleration or override signal.

- Interrupt type custom macro

(1) Do not initiate any interrupt during reverse execution.
(2) Do not execute an interrupted block and the interrupt program in reverse execution.

- Tool management function

The tool life is not counted during reverse execution and forward reexecution.

- Inverse time feed (G93)

If a nonzero value is set as the feedrate to be applied during reverse execution in parameter No. 1414, a block that moves the tool by inverse time feed during forward execution is executed at the parameter-set feedrate (feed per minute) during reverse execution.

If the feedrate during reverse execution (parameter No. 1414) is not set (= 0), the same feedrate as applied during forward execution is used.

- Maximum spindle speed clamp (G92Sxxxx)

Clamping at a maximum spindle speed specified during reverse execution becomes valid. This means that if G92Sxxxx appears during reverse execution, the spindle speed is clamped at Sxxxx in the subsequent reverse execution. As a result, the clamp speed may differ between reverse execution and forward execution even when the same block is executed. The spindle speed is clamped when the G96 mode is set.

- Auxiliary functions

M, S, T, and the second auxiliary function (B function) are output directly also during reverse execution and forward reexecution.
When specified together with a move command in the same block, M, S, T, and the second auxiliary function (B function) are output with the move command at the same time during forward execution, reverse execution, and forward reexecution. Therefore, the output positions of M, S, T, and the second auxiliary function (B function) during reverse execution differ from those during forward execution and forward reexecution.

- Custom macro operation

Custom macro operations are ignored during reverse execution and forward reexecution.

- Tool retract and recover function

For retract operation and repositioning operation by the tool retract and recover function, reverse execution cannot be performed. Retract operation and repositioning operation are ignored during reverse execution and forward reexecution.

- AI contour control

During reverse execution and forward reexecution, the feedrate clamp function by acceleration under AI contour control is disabled.

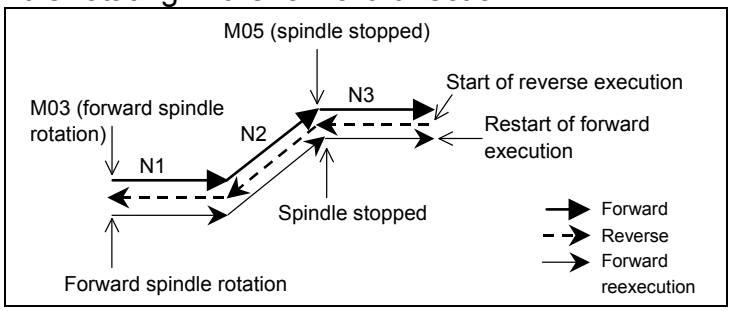
- Display

During reverse execution and forward reexecution, the modal display and the display of the currently executed program are not updated; information obtained at the start of reverse execution is maintained.

Warning

⚠ WARNING

1 Auxiliary functions are output directly even during reverse execution and forward reexecution. Accordingly, the execution status of an auxiliary function during forward execution may be reversed during reverse execution.
Example:
When forward rotation of the spindle (M03) and stop (M05) are specified
When N3 is executed during reverse execution, M05 is output. So, when N2 and N1 are executed during reverse execution, operation is performed with the spindle stopped.
When N1 is executed during forward reexecution, M03 is output. So, when N1 and N2 are executed during forward reexecution, operation is performed with the spindle rotating in the forward direction.



2 To perform reverse execution after a feed hold stop or single block stop operation, be sure to restore the original position if manual intervention has been performed after the stop , then set the "REVERSE" switch to on. Movements made by manual intervention are ignored during reverse execution and forward reexecution. (The same operation as in the manual absolute off state takes place.)
If manual intervention is performed during reverse execution or forward reexecution, the amount of manual intervention is added to the coordinate system at a restart after a stop due to a feed hold or single block during forward execution after the end of forward reexecution. Whether to add the amount of manual intervention follows the manual absolute switch.

4.14 ACTIVE BLOCK CANCEL FUNCTION

The automatic operation can be stopped by inputting block cancellation signal BCAN while automatic operation. After an automatic operation becomes stop, automatic operation signal OP and cycle start lamp signal STL becomes "0".
All modal information is preserved.
At this time,

(1) If the machine is moving, it stops while decelerating. And the distance remaining is cleared.
(2) If the dwell is executing, it is canceled.
(3) If auxiliary function, spindle speed, tool function, the second auxiliary function is executing, the each strobe signal MF, SF, TF, BF becomes "0" and the block is ended.
If the high speed M/S/T/B interface (bit 7 (MHI) of parameter No.3001 = 1) is used, the block is ended without the each strobe signal MF, SF, TF and BF changing.
Therefore, the completion signals MFIN, SFIN, TFIN, and BFIN must be set to the same logical level as the corresponding strobe signals before the operation is restarted.

An automatic operation is restarted from the next block of the canceled block by cycle start.

Active block cancel function

Explanation

Operation when operation restarts

G90/G91

When the operation is restarted, the operation of the restart is decided depending on the command in the next block of the canceled block.

If the next block is an absolute command (G90), the operation is restarted from the position at which the operation is stopped, toward the terminal of the next block. When the next block is incremental command (G91), the incremental operation is restarted from the stopping position.

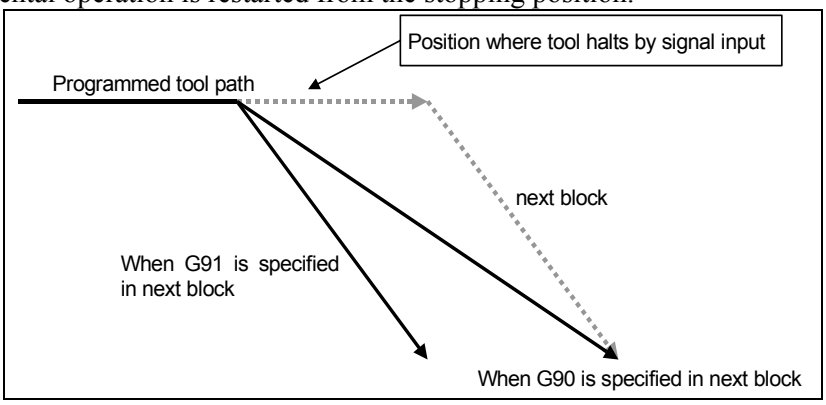


Fig. 4.14 (a) Absolute / Incremental

> **⚠ CAUTION**
> 1 Because the tool becomes a start from the canceled position when an automatic operation restarts, the route of an actual movement is different from original program route. Therefore, there is a possibility that the tool route and the work piece interfere at the restart. Please do the restart operation after confirming neither the tool nor work piece interfere.
> 2 If the program uses incremental commands, the terminal of the next block shifts by an amount corresponding to the previous cancellation when automatic operation restarts. This shift remains until the absolute command is executed next. Therefore, this function cannot be used to check the block terminal by the program made by the incremental command.

Tool radius / tool nose radius compensation

The offset mode is temporarily canceled at that time when canceling in the tool radius / tool nose radius compensation mode. And, the start-up operation is done at the restart.

For example, when canceling in N20 of the undermentioned program, the movement of restarting is the start-up operation toward the position commanded with N30 in the figure below.

Example

```
N10 G90 G01 G42 X_ D_ S_ F_ ;
N20 X_ ;
N30 X_ Y_;
N40 X_ ;
```

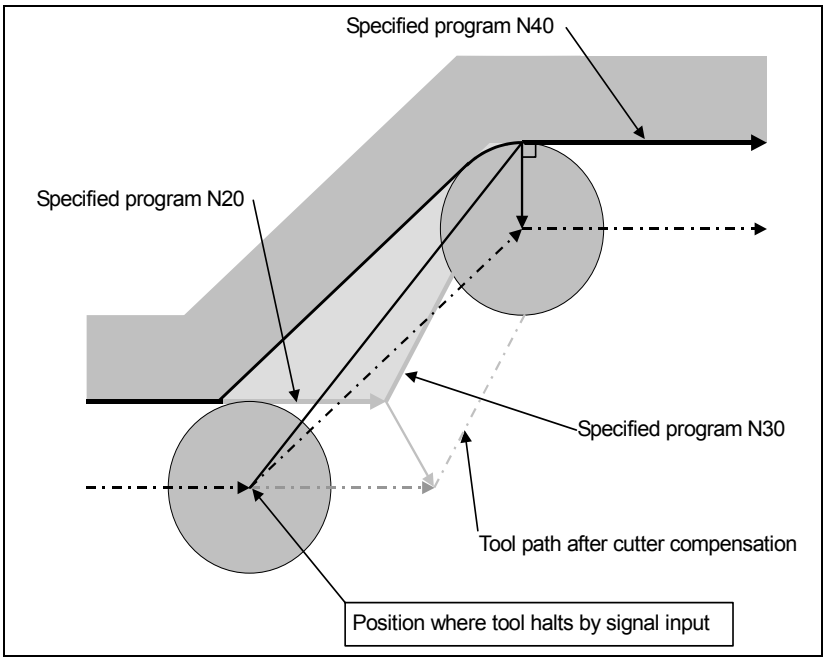


Fig. 4.14 (b) Tool radius / tool nose radius compensation

Canned cycle for drilling

When a block is canceled during a canned cycle for drilling, that canned cycle is canceled. Program execution is restarted from the next block.

Example1
```
N10 G91 G84 X_ Y_ Z_ R_ K3 ;
N20 G90 G00 X_ Y_ Z_ ;
```

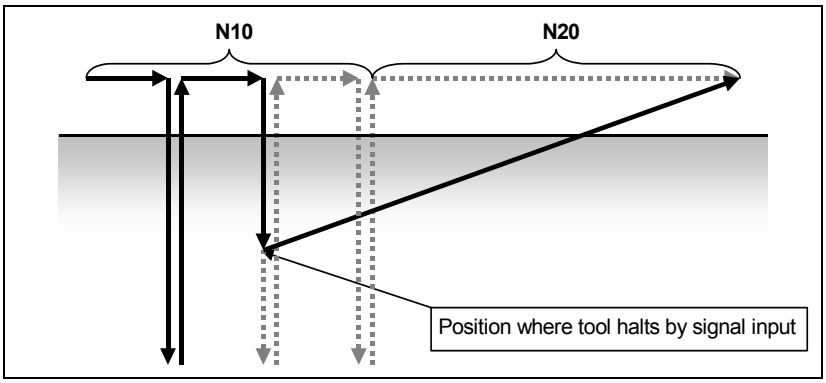


Fig. 4.14 (c)Canned cycle for drilling 1

Example 2
```
N10 G90 G84 X_ Y_ Z_ R_ ;
N20 X_ Y_ ;
N30 X_ Y_ ;
N40 X_ Y_ ;
N50 G00 X_ Y_ Z_ ;
```

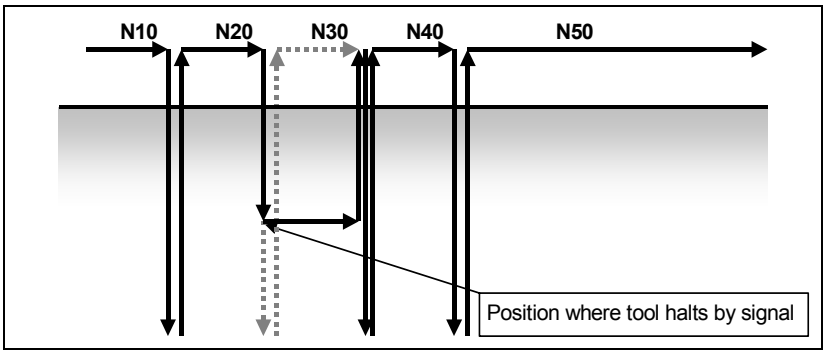


Fig. 4.14 (d) Canned cycle for drilling 2

At the restart, the operation is executed from the next N30 block when the operation is canceled while executing in N20 cycle.
At this time, because the G84 mode is maintained, the tool positions in R point after moving to the X-Y position commanded with N30 and starts the hole opening operation.

Notes

⚠ CAUTION

1 Please confirm neither the tool route nor work piece interference when the operation is restarted.
2 When an automatic operation is restarted, the route of an actual movement is different from an original route because it is a start from the canceled position. Therefore, there is a possibility that the tool route and the work piece interference at the restart.
3 When reset or the emergency stops is executed during the stopping by this function, this function becomes invalid. In a word, maintained various data and modal data are changed to the clear or reset states by the reset operation.
4 This function is effective only while automatic operation. (OP=1 and STL=1)
5 If the part program is incremental command, the terminal of the next block shifts cancellation amount when automatic operation restarts. This amount of the gap keeps remaining until the next absolute command is executed. Therefore, this function cannot be used to check the block terminal by the program made by the incremental command.
6 This function is not available during AI contour control mode. Even if the cancel signal is input, it is ignored.

5 TEST OPERATION

The following functions are used to check before actual machining whether the machine operates as specified by the created program.

5.1 MACHINE LOCK AND AUXILIARY FUNCTION LOCK....1105
5.2 FEEDRATE OVERRIDE....1106
5.3 RAPID TRAVERSE OVERRIDE....1107
5.4 DWELL/AUXILIARY FUNCTION TIME OVERRIDE....1107
5.5 DRY RUN....1109
5.6 SINGLE BLOCK....1110
5.7 HIGH SPEED PROGRAM CHECK FUNCTION....1111
5.8 MANUAL HANDLE RETRACE....1113
5.9 AUXILIARY FUNCTION OUTPUT BLOCK REVERSE MOVEMENT FOR MANUAL HANDLE RETRACE....1126
5.10 MANUAL HANDLE RETRACE FUNCTION FOR MULTI-PATH....1126
5.11 EXTENSION OF THE MANUAL HANDLE RETRACE FUNCTION....1130

5.1 MACHINE LOCK AND AUXILIARY FUNCTION LOCK

To display the change in the position without moving the tool, use machine lock.
There are two types of machine lock: all-axis machine lock, which stops the movement along all axes, and specified-axis machine lock, which stops the movement along specified axes only. In addition, auxiliary function lock, which disables M, S, T, and B (2nd auxiliary function) commands, is available for checking a program together with machine lock.

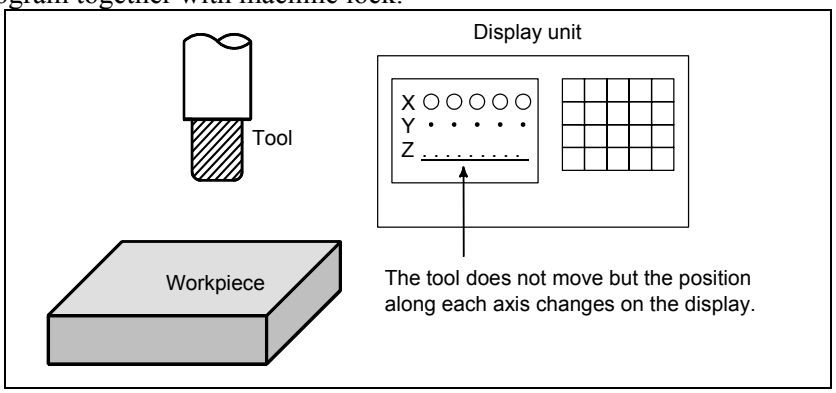


Fig. 5.1 (a) Machine lock

Machine lock and auxiliary function lock

Procedure

- Machine Lock

Press the machine lock switch on the operator's panel. The tool does not move but the position along each axis changes on the display as if the tool were moving.
Some machines have a machine lock switch for each axis. On such machines, press the machine lock switches for the axes along which the tool is to be stopped. Refer to the appropriate manual provided by the machine tool builder for machine lock.

> ⚠ **WARNING**
> The positional relationship between the workpiece coordinates and machine coordinates may differ before and after automatic operation using machine lock. In such a case, specify the workpiece coordinate system by using a coordinate setting command or by performing manual reference position return.

- Auxiliary function lock

Press the auxiliary function lock switch on the operator's panel. M, S, T, and B codes are disabled and not executed. Refer to the appropriate manual provided by the machine tool builder for auxiliary function lock.

Limitation

- M, S, T, B command by only machine lock

M, S, T and B commands are executed in the machine lock state.

- Reference position return under machine lock

When a G27, G28, or G30 command is issued in the machine lock state, the command is accepted but the tool does not move to the reference position and the reference position return LED does not go on.

- M codes not locked by auxiliary function lock

M00, M01, M02, M30, M98, M99, and M198 (subprogram call function) commands are executed even in the auxiliary function lock state. M codes for calling a subprogram (parameters Nos. 6071 to 6079) and those for calling a custom macro (parameters Nos. 6080 to 6089) are also executed.

5.2 FEEDRATE OVERRIDE

A programmed feedrate can be reduced or increased by a percentage (%) selected by the override dial. This feature is used to check a program.
For example, when a feedrate of 100 mm/min is specified in the program, setting the override dial to 50% moves the tool at 50 mm/min.

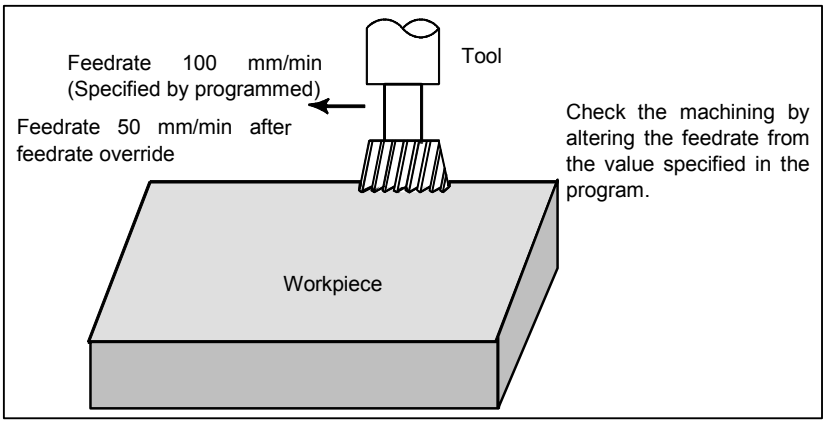


Fig. 5.2 (a) Feedrate override

Feedrate override

Procedure

Set the feedrate override dial to the desired percentage (%) on the machine operator's panel, before or during automatic operation.
On some machines, the same dial is used for the feedrate override dial and jog feedrate dial. Refer to the appropriate manual provided by the machine tool builder for feedrate override.

Limitation

- **Override range**

The override that can be specified ranges from 0 to 254%. For individual machines, the range depends on the specifications of the machine tool builder.

- **Override during thread**

During the threading process, the override setting is ignored; it is always regarded as 100% during the process.

5.3 RAPID TRAVERSE OVERRIDE

An override of four steps (F0, 25%, 50%, and 100%) can be applied to the rapid traverse rate. F0 is set by a parameter No. 1421.

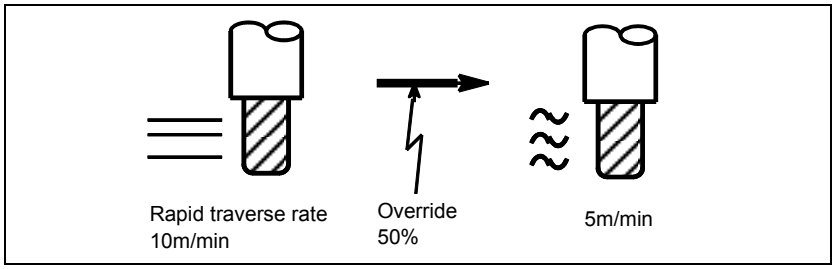


Fig. 5.3 (a) Rapid traverse override

Rapid traverse override

Procedure

Select one of the four feedrates with the rapid traverse override switch during rapid traverse. Refer to the appropriate manual provided by the machine tool builder for rapid traverse override.

Explanation

The following types of rapid traverse are available. Rapid traverse override can be applied for each of them.

(1) Rapid traverse by G00
(2) Rapid traverse during a canned cycle
(3) Rapid traverse in G27, G28, G29, G30, G53
(4) Manual rapid traverse
(5) Rapid traverse of manual reference position return

5.4 DWELL/AUXILIARY FUNCTION TIME OVERRIDE

An override can be applied to dwell and the auxiliary function (M/S/T/B) in increments of 1% in the range from 0 to 100%.

For the auxiliary function (M/S/T/B), however, if the override is less than 100%, the next block is started after a lapse of the actual processing time plus the shortage time.

Dwell/auxiliary function time override

Procedure

Press the 1% rapid traverse override selection switch and turn the 1% rapid traverse override switch to select the desired percentage.

For details on 1% rapid traverse override, refer to the manual provided by the machine tool builder.

Explanation

- Dwell

An override can be applied to dwell.

[Example]

Override and dwell time when G04 P10000; is executed

Table 5.4 (a)

Override	Dwell time
100%	10.0 sec.
75%	13.3 sec.
50%	20.0 sec.
25%	40.0 sec.

NOTE
An override is disabled for dwell per revolution.

- AUXILIARY functions (M/S/T/B)

An override can be applied in the time from when M/S/T/B is sent until the FIN signal is turned off (or until MFIN, SFIN, TFIN, and BFIN are changed in the case of the high speed M/S/T/B). The time required to communicate with PMC is included.

When the FIN signal is turned off and then shortage time elapses, the next block is started.

[Example]

When the following program is executed with the override set to 60%

N10 M03 ;

N20 · · · · · ;

When the processing time of the N10 block is 80 msec, the N20 block is started approx. 56 msec after completion of the processing.

Shortage time = 80 msec × 100/60 - 80 msec = 53.333 ≒ 56 msec

NOTE
The time count is incremented every 4 msec or 8 msec and the fraction is rounded up. The counting cycle of time may depend on the system.

The 1% rapid traverse override signal reads the value upon completion of the auxiliary function. Accordingly, the value is not ensured if the override value changes between before and after the reading.

When the movement command and auxiliary function are specified in the same block:

(1) When the movement command is completed first
- An override is enabled.

(2) When the auxiliary command is completed first
- An override is disabled.

When auxiliary functions are specified in the same block, the shortage time is calculated using the auxiliary function with the longest processing time.

NOTE
1 This function is disabled for dwell by PMC axis control (04H) and the auxiliary functions (12H, 14H, 15H).
2 For the dwell, when the 1% rapid traverse override signal is 0%, a transition to the next block is not performed.
3 The time when a feed hold is caused in the auxiliary function during shortage time is not included in the shortage time.

NOTE
4 For the auxiliary function, when the 1% rapid traverse override signal is 0%, the setting of parameter No. 3018 is used as the override value. When the setting of the parameter is 0, it is assumed to be 10%. While the 1% rapid traverse override signal is 0%, a transition to the next block is not performed.
5 For auxiliary functions M02 and M30, bit 7 (OVM) of parameter No. 3002 can be used to enable or disable an override.

5.5 DRY RUN

The tool is moved at the feedrate specified by a parameter regardless of the feedrate specified in the program. This function is used for checking the movement of the tool under the state that the workpiece is removed from the table.

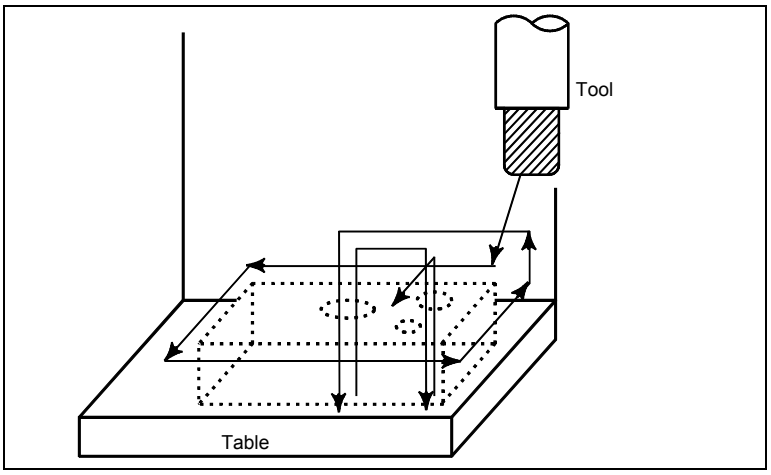


Fig. 5.5 (a) Dry run

Dry run

Procedure

Press the dry run switch on the machine operator's panel during automatic operation.
The tool moves at the feedrate specified in a parameter. The rapid traverse switch can also be used for changing the feedrate.
Refer to the appropriate manual provided by the machine tool builder for dry run.

Explanation

- Dry run feedrate

The dry run feedrate changes as shown in the Table 5.5 (a) according to the rapid traverse switch and parameters.

Table 5.5 (a)

Rapid traverse switch	Program command	
	Rapid traverse	Cutting feed
ON	Rapid traverse rate	Dry run feedrate × Jvmax(*2)
OFF	Dry run feedrate × JV, or rapid traverse rate (*1)	Dry run feedrate × JV(*2)

Max. cutting feedrate.....Setting by parameter No.1430
Rapid traverse rate.....Setting by parameter No.1420
Dry run feedrate .....Setting by parameter No.1410

(*1) Dry run feedrate × JV when bit 6 (RDR) of parameter No. 1401 is 1. Rapid traverse rate when parameter RDR is 0.
JV Jog feedrate override

(*2) Clamped to the maximum cutting feedrate
Jvmax Maximum value of jog feedrate override

5.6 SINGLE BLOCK

Pressing the single block switch starts the single block mode. When the cycle start button is pressed in the single block mode, the tool stops after a single block in the program is executed. Check the program in the single block mode by executing the program block by block.

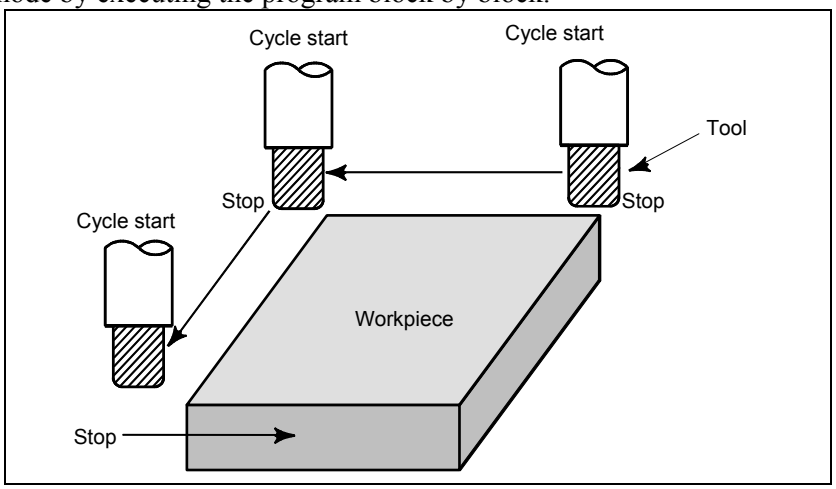


Fig. 5.6 (a) Single block

Single block

Procedure

1 Turn the single block switch on the machine operator's panel on.
2 Press the cycle start button. The machine executes one block and stops.
3 Press the cycle start button again. The machine executes the next one block and stops.
Refer to the appropriate manual provided by the machine tool builder for single block execution.

Explanation

- Reference position return and single block

If G28, G29, and G30 are issued, the single block function is effective at the intermediate point.

- Single block during a canned cycle

In a canned cycle, the single block stop points are the end of <1>, <2>, and <6> shown below. When the single block stop is made after the point <1> or <2>, the feed hold LED lights.

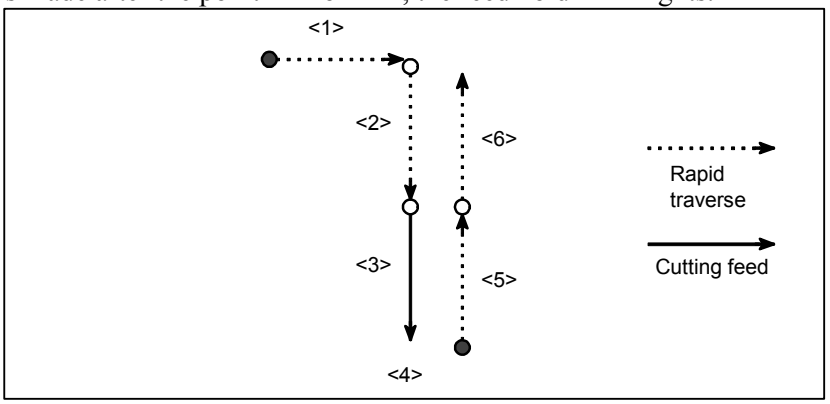


Fig. 5.6 (b) Single block during canned cycle

- Subprogram call and single block

Single block stop is not performed in a block containing M98P_;. M99; or G65.
However, single block stop is even performed in a block with M98P_ or M99 command, if the block contains an address other than O, N, P, L.

5.7 HIGH SPEED PROGRAM CHECK FUNCTION

When the cycle start button is pressed with the high speed program check mode set to be enabled, the program syntax and stroke limit are checked without axis movement.
A program check is performed at the maximum speed that can be specified by the CNC, regardless of the specified speed.

High speed program check

Procedure

1 Perform position of the machine at the start position of the machining program in advance.
Set the workpiece coordinate system or tool compensation to the same state as in actual machining.
2 Reset the CNC and press the high speed program check switch on the machine operator's panel to enter the high speed program check mode. In the high speed program check mode, "PRG-CHK" blinks at the lower-left of the CNC screen.
3 Press the cycle start button to execute the program without axis movement. The feedrate is the maximum speed that can be specified by the CNC, regardless of the specified speed.
The program syntax and stroke limit are checked during execution of the program.
4 Upon completion of the check, reset the CNC and release the high speed program check mode with the switch on the machine operator's panel.

Explanation

- Stroke Limit Check

Following the stroke limit check is executed without axes movements during the high speed program check mode.

- Stored stroke check 1
- Stored stroke check 2, 3
- Stroke limit check before move
- Rotation area interference check

If an alarm about the stroke limit occurs in the high speed program check mode, clear the alarm as described below.
(1) Change the mode to other than MEM or DNC.
(2) Reset the CNC.

> ⚠ **WARNING**
> 1 If the machine coordinate system is not established, a stroke limit check is not performed correctly. When a reference position return is completed after power-on, perform a high speed program check.
> 2 When the width of the prohibited area is small in a stored stroke limit 2 and 3 check and rotation area interference check, if a command to intersect the prohibited area in one block is issued, the stroke limit may not be detected. This means that a program not causing a stroke check alarm in a high speed program check may cause an alarm in actual operation.

- Program Check Feedrate

During the high speed program check mode, the feedrate of program execution becomes the maximum feedrate of CNC system regardless of the specified feedrate.

The feedrate clamp, override and dry run are not effective.
The execution speed of circular interpolation, involute interpolation, spiral/conical interpolation, and NURBS interpolation can be selected by bit 7 (PGF) of parameter No. 1490 from the maximum speed and dry run speed.

NOTE
1 The execution time of dwell is the same as a normal operation.
2 The alarm PS011, "FEED ZERO (COMMAND)" occurs when there is no F command in a normal operation. However, it is executed at maximum speed without the alarm in the high speed program check mode even if there is no F command.

- Workpiece Coordinate System Preset

Upon completion of the high speed program check mode, the workpiece coordinate system is automatically preset. This enables program operation without making a reference position return upon completion of a program check. For details, refer to the manual provided by the machine tool builder.

⚠ WARNING
When the coordinate system is set before the high speed program check mode is turned on by the work coordinate system setting G92 (machining center system and G code system B or C of the lathe system), G50 (G code system A of the lathe system) and the local coordinate system setting G52, their setting values are deleted by the execution of the work coordinate system preset.

- Machine Coordinate Display for Program Check

During the high speed program check mode, it is possible to change the display of machine coordinates from the actual machine coordinates to the machine coordinates for the program check by setting bit 7 (PGM) of parameter No.11320.
And it is possible to change the display of machine coordinate for each axis from the actual machine coordinate to the machine coordinate for the program check by setting bit 4 (PGA) of parameter No.3115.

- Tool life

If tool life management command or tool management command is specified during the high speed program check mode, tool life is not counted.

- Unavailable command

Following functions can not be specified during the high speed program check mode. If the following functions are specified, the alarm PS5364, "ILLEGAL COMMAND IN PROGRAM CHECK" occurs.

- High speed cycle cutting
- Chopping
  As for the chopping by the input signal:
  The alarm PS5364 occurs when the program check is started during the chopping by the input signal.
  The alarm PS5364 occurs when the chopping by the input signal is started at the program check execution.
- Electronic gear box

▪ Commands not covered by a high speed program check

Auxiliary functions and other commands are not covered by a high speed program check. For details, refer to the manual provided by the machine tool builder.

5.8 MANUAL HANDLE RETRACE

Overview

In this function, the program can be executed both forward and backward with a manual handle (manual pulse generator) under automatic operation.
Therefore, errors of a program, interference, and so on can be checked easily by working a machine actually.

- Checking mode

In this mode, the program can be executed forward and backward and the program can be checked.
To change to the checking mode, it is necessary to change the mode to MEM mode, and the checking mode signal MMOD is set to "1". This function makes the data to execute the program backward when the program is executed forward in the checking mode.
To work a machine synchronizing with a pulse generated by a manual handle in the checking mode, the manual handle check signal MCHK is set to "1" in addition to the above-mentioned. As a result, it becomes possible to check the program with a manual handle.

NOTE
During the checking mode, it is not possible to change the parameter and offset.

- Forward movement with a manual handle

The "forward movement" is that the program is executed forward by turning a manual handle in the positive direction (when the manual handle check signal is set to "1".) or in no relation to rotation of a manual handle (when the manual handle check signal is set to "0".).
When the manual handle check signal is set to "1", the execution speed of the program is proportional to the number of rotations of a manual handle. The program is executed forward rapidly when a manual handle is turned to the positive direction rapidly. And, the program is executed forward slowly when a manual handle is turned to the positive direction slowly. The distance magnification traveled per pulse from manual handle can be switched as same as a usual manual handle feed function.
When the manual handle check signal is set to "0", the execution of the program is controlled as same as an automatic operation.

- Backward movement

The "backward movement " is that the program executed forward once is executed backward by turning a manual handle in the negative direction.
The program can be executed backward only for the block executed forward. And, the number of blocks for it is about 190 blocks. This block number changes by the content of the commanded program.
The program is executed backward rapidly when a manual handle is turned to the negative direction rapidly. And the program is executed backward slowly when a manual handle is turned to the negative direction slowly. The distance traveled per pulse from manual handle can switch magnification as well as a usual manual handle feed.

NOTE
In the system which exceeds 4 paths and 20 axes, the maximum number of blocks which can move backward becomes about 97 blocks.

Explanation

- Control by the manual handle

Program execution start

The checking mode signal MMOD is set to "1" in MEM mode in order to change the checking mode. Then, the program execution is begun by turning ST signal from "1" to "0".

If the manual handle check signal MCHK is set to "1" at this time, the execution of the program is controlled by a manual handle. The program is executed synchronizing with rotation of a manual handle.
When a manual handle check signal MCHK is set to "0", it is controlled as usual execution.
When check mode signal MMOD is set to "1" during the operation of the program, it is enabled a check mode from the block that next buffering is done.
That is, even if check mode signal set to "1", check mode is not always enabled at once.
When check mode is enabled, check mode confirmation signal MMMOD is set to "1".

NOTE
After the signal MMOD is turned to "0" during the execution of the program, the program cannot be executed forward and backward.

Control with the manual handle

The value of the parameter No.6410 and the scale factors decide the moving speed of the machine by one pulse generated by a manual handle.
When a manual handle is turned, the actual movement speed of the machine is as follows.
[Feedrate command value] × [Number of the handle pulse per a second]× [Handle magnification]
× ([Parameter setting value]/100) × (8/1000) (mm/min or inch/min)
Example) When feedrate command value is 30mm/min, handle magnification is 100, parameter No.6410 is set to 1 and manual pulse generator is rotated at 100 pulse/rev, the feedrate of axis is decided as follow.
[Feedrate]=30[mm/min] × 100[pulse/s] × 100 × (1/100) × (8/1000)[s] =24[mm/min]

When the feedrate exceeds the override 100% feedrate by turning a manual handle rapidly, the feedrate is clamped at the speed of override 100%. That is, if the pulse of the following formula exceeds "1", the feedrate is clamped.
[Number of the handle pulse per a second]
× [Handle magnification] × ([Parameter setting value]/100) × (8/1000)
The rapid traverse feedrate is clamped at 10%. However, the feedrate of the rapid traverse is clamped at 100% when bit 0 (HDRPD) of parameter No.6400 is set to "1".
And if parameter No.6405 is set to an optional value, it can be clamped to override by nearly optional value.
When the parameter No.6405 is set to larger value than "100", it is clamped to nearly 100%.
When parameter No.6405 is set to "0", the setting of bit 0 (RPO) of parameter No.6400 becomes valid.
The single block signal and the feed hold signal are effective in the checking mode. When the execution of a program is stopped by the single block stop or the feed hold stop, it is necessary to turn ST signal from "1" to "0" in order to restart the program.
In the block with the movement and the block of dwell, the execution speed of the program can be controlled by turning a manual handle. As for the block of neither movement nor dwell such as the block of only address M, S, T, and F, the program advances to the following block even if a manual handle does not turn.
The rotation of the spindle does not synchronize with a pulse of a manual handle. During the checking mode, the spindle rotates at the specified rotation speed. As for the feed per revolution, a program is executed at the feedrate which was converted from the rotation speed of the spindle to the corresponding feed per minute inside CNC.

NOTE
The manual handle used by this function is always the first. The 2nd and 3rd manual handle cannot be used in this function.

Forward movement and backward movement with a manual handle

The program is executed forward when a manual handle is turned to the positive direction. And, the program is executed backward when a manual handle is turned to the negative direction.

The program is executed backward as soon as a manual handle is turned to the negative direction in executing the program forward.
When a manual handle keeps being turned in a negative direction, the program is executed backward and the execution stops in the block of O number. Then, if a manual handle is turned to the positive direction, the program is executed forward again.
Even if a manual handle controls the program execution, the program is executed forward in no relation to a pulse generated by a manual handle on setting the manual handle check signal to "0".

Program end

When the block of M2 or M30 is executed, the manual handle retrace ends. It is not possible to execute the program backward from the block of M2 or M30.
When the execution of the program ends, RESET signal must be set to "1", and the checking mode signal and the manual handle check signal must be set to "0".
In 2 path control system, FIN signal must not be set to "1" when the block of M2 or M30 is executed in only one of paths. After the block of M2 or M30 has been executed in both paths, FIN signal is set to "1". (Except for the block of waiting M code is commanded before M2 or M30 in both paths.)

- Notice of the operation

- Dry-run can not operate during the checking mode. Dry-run signal must be set to "0".
- Automatic operation starts immediately with the feedrate commanded by the program, when the checking mode signal or the synchronous operation with handle signal is turned off during executing the program in the checking mode.
- The edit of the program and the change of the parameter and the offset must not be done.
- While the program is being executed backward or blocks once executed backward are being executed forward again, do not change the checking mode signal MMOD.

- Backward movement of each code

All modal information of G, T, S-code is memorized in executing the program forward. And, the memorized data of the modal G, T, S-code are used in executing the program backward.
As for M-codes, they are grouped and the modal information is managed by parameter No.6411 to 6490. Therefore, M-code can be executed backward according to the information. As for the modal information of the M-code, a change in each group is memorized in the execution data.
As for the codes except for G, M, S, and T, the same code is output between forward movement and backward movement.

- G-code

If G-code that changes modal information is commanded in backward movement, the modal information of previous block is executed.
Example)

```
N1G99;
N2G01X_F_;
N3X_Z_;
N4G98;  ..... backward movement starts from this block
N5X_Y_Z_;
```

If backward movement starts from N4 block, the modal information is changed from G98 to G99 and G99 is executed from N3.
G-code with a movement is traced along the route opposite to forward movement.
G-code that can be command in executing the program backward is as follows.
The other G-codes cannot be command in executing the program backward.
The G-codes in the G-code system B and C also can be used.

Lathe system

G00	G01	G02	G03	G04	G22	G23
G25	G26	G28	G30	G40	G41	G42
G50	G53	G65	G70	G71	G72	G73
G75	G80	G83	G85	G87	G89	G90
G94	G96	G97	G98	G99		

(G-code system A)

Machining center system

G00	G01	G02	G03	G04	G22	G23
G25	G26	G28	G30	G40	G41	G42
G43	G44	G49	G53	G65	G73	G76
G80	G81	G82	G83	G85	G86	G87
G88	G89	G82	G94	G95	G96	G97

NOTE
1 In Small-Hole Pecking Drilling Cycle(G83), backward movement is prohibited.
2 In forward movement of Boring Cycle(G88), the sequence of actions at bottom of hole is shown as follows (dwell -> stop of spindle motor -> hold state). But in backward movement, that is (rotation of spindle -> hold state -> dwell after restart).

- M-code

If there is M-code in the same group is commanded in previous blocks, modal information of the M-code, commanded at the last in previous blocks, is output.
If no M-code is commanded in previous blocks, the M-code set to the first parameter in the same M-code group is output.
If M-code is not set to group M-code in parameter, the same M-code is output in backward movement.
If the bit 5 (RVN) of parameter No. 6400 is set to "1", the backward movement is prohibited when the M-code, which is not set to group M-code, is commanded in backward movement.

NOTE
When setting the parameter RVN, backward movement prohibition is enabled except the M-code which was set in the grouping but backward movement can be enabled for the following M-code exceptionally.
1. Subprogram Call by M98/M99.
2. Subprogram Call using an M code
3. Macro Call using an M code
4. Waiting M code
5. M0

Example)

Output of M-codes that are set to groups by parameters in backward movement

Setting of parameters:

Bit 2 (MC5) of parameter No.6400=1 and bit 3 (MC8) of parameter No.6400=0 (5 M-codes/group and 16 groups)

No.6411=100
No.6412=101
No.6413=102
No.6414=103
No.6415=104
} Group A

No.6416=200
No.6417=201
No.6418=202
No.6419=203
No.6420=204
} Group B

Program O10 is executed in forward movement from N1 to N15 and backward movement is executed from N15. In backward movement, the output of M-codes is shown as next table.

Table 5.8 (a)

	Forward movement	Backward movement
O10;		
N1G4X1.;		
N2M101;	M101	M100 (*1)
N3G4X1.;		
N4M204;	M204	M200 (*1)
N5G4X1.;		
N6M104;	M104	M101 (*2)
N7G4X1.;		
N8M300;	M300	M300
N9G4X1.;		
N10M200;	M200	M204 (*1)
N11G4X1.;		
N12M0;	M0	M0 (*3)
N13G4X1.;		
N14M102;	M102	M104 (*2)
N15G4X1.;	Backward movement starts from this block	
M2;		

*1 No M-code in the same group is commanded before this block, so the M-code, which is set in the 1st. parameter of the same group, is output.

*2 M-code in the same group is commanded before this block, so the M-code, which is commanded at the last before this block, is output.

*3 M-code is not set to group M-code, so the same M-code is output.

- S and T-code

A modal value of the previous block is output.

When movement command and S-code or T-code is commanded in the same block, the timing of the output of the S-code and T-code is different. Because, the timing where S-code and T-code are output at the forward movement is different from that at the backward movement. By setting bit 7 (STO) of parameter No.6401 to "1", the timing of the output of S and T code at the forward movement is the same as the one at the backward movement.

Example)

T-code output timing at the backward movement

T-code is output as follows when the program proceeds backward after the forward movement to N8 block.

Table 5.8 (b)

	Forward movement	Backward movement	
		Parameter STO=0	Parameter STO=1
O1000;			
N1G98G00X0Z0;			Default T output
N2G00X-10.T11;	T11 output	Default T output	
N3G00X100.;			T11 output
N4G00X10.Z20.T22;	T22 output	T11 output	No T code output
N5G00X30.Z30.;			
N6G00X-10.Z-20.;			T22 output
N7G00X50.Z40.T33;	T33 output	T22 output	T33 output
N8G04X5.;		(Backward start)	(Backward start)
M30;			

The “Default T” means a T-code status at N1 block in forward movement. If the status is T0, “T0” signal is output as “Default T” in the backward movement.

The timing of T-code output of N7 and N8 in O1000 shown in the example above is as follows.

Forward movement :



Backward movement (When parameter STO is set to “0”) :



Backward movement (When parameter STO is set to “1”) :



Fig. 5.8 (a)

- Direction change prohibition

The direction change prohibition is a state not changing the direction where the program is executed.
In the state, even if the rotating direction of a manual handle is reversed, the reversed rotation is ignored.
A manual handle must be rotate in the same direction as present direction for removing this state.
The direction change prohibition can be confirmed by output signal MNCHG.

It becomes the change prohibition state under the following condition.

- While the block with the code waiting for FIN is executing
- After a block has done and until the next block begins to operate
- During thread cutting
- Modal G code of G68.1, G68 and G51.2
- The block with the axis that ends movement earlier in the block with G02 or nonlinear type position (G00) etc.
- During time waiting at the switching of blocks (for multi-path systems only. See "Time waiting in multi-path systems".)

- Backward movement prohibition

The backward movement prohibition is a state that the program cannot be executed from a certain block backward. In this state, the negative rotation of a manual handle is ignored, and the only positive rotation is effective. The program must be executed forward by rotating a manual handle in the positive direction for removing this state.
The backward movement prohibition can be confirmed by output signal MRVSP.
If the following blocks are executed in backward movement, backward movement is prohibited.

- Program number block of main program (except subprogram and macro program)
- Over the maximum number of the blocks for reverse movement
- The block including backward movement prohibition G-code (which is not described in the paragraph "G-code")
- The block which is executed while in modal including backward movement prohibition G-code (which is not described in the paragraph "G-code")

- Status display

In manual handle retrace, the status of manual handle retrace is displayed on clock display of CNC state display line. This status display is displayed during the execution of manual handle retrace. The clock is displayed usually.

When the all condition is filled, "M.H.RTR." is displayed on clock display of CNC state display line. This status is displayed by the color of color number 3 (INPUT KEY, O/N NO. and STATUS are the same color). The screen display is as shown in Fig. 5.8 (b). When the following conditions are not full, the clock is displayed.

- When bit 2 (CHS) of parameter No.6401 is set to "0":
  1) Software option of handle manual retrace is enabled.
  2) Status display disable/enable bit 6 (HST) of parameter No.6401 is set to "1".
  3) Check mode confirmation signal MMMOD is set to "1".

- When bit 2 (CHS) of parameter No.6401 is set to "1":
  1) Software option of handle manual retrace is enabled.
  2) Status display disable/enable bit 6 (HST) of parameter No.6401 is set to "1".
  3) Cycle start signal STL is set to "1".
  4) Check mode signal MMOD is set to "1".
  5) Manual handle check signal MCHK is set to "1".

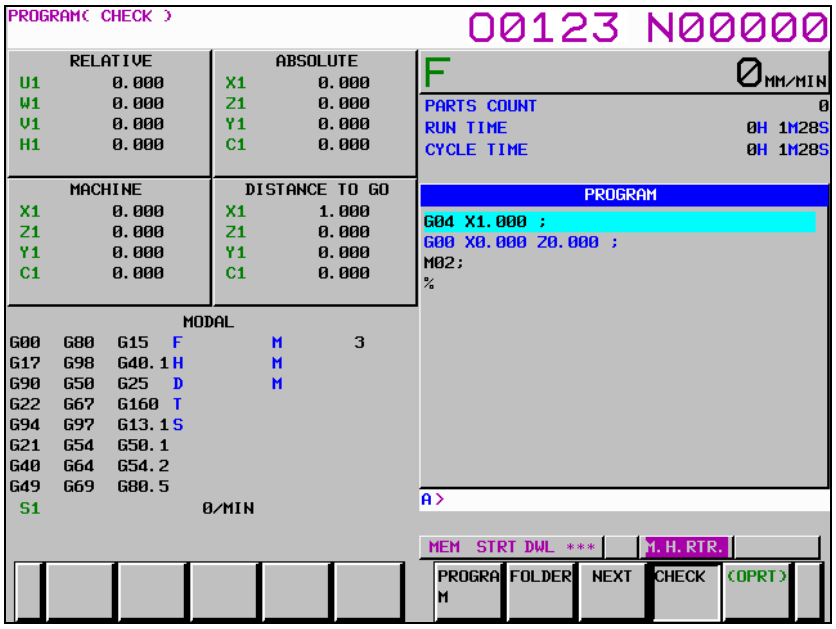


Fig. 5.8 (b) "M.H.RTR." status display

Besides, when reverse movement prohibition signal MRVSP is set to "1",
the "NO RVRS." is displayed. This status is displayed by blinking/reversing in the color of color number 1 (ALARM is the same color). The screen display is as shown in Fig. 5.8 (c). When reverse movement prohibition signal MRVSP is set to "0", the " M.H.RTR." is displayed again.

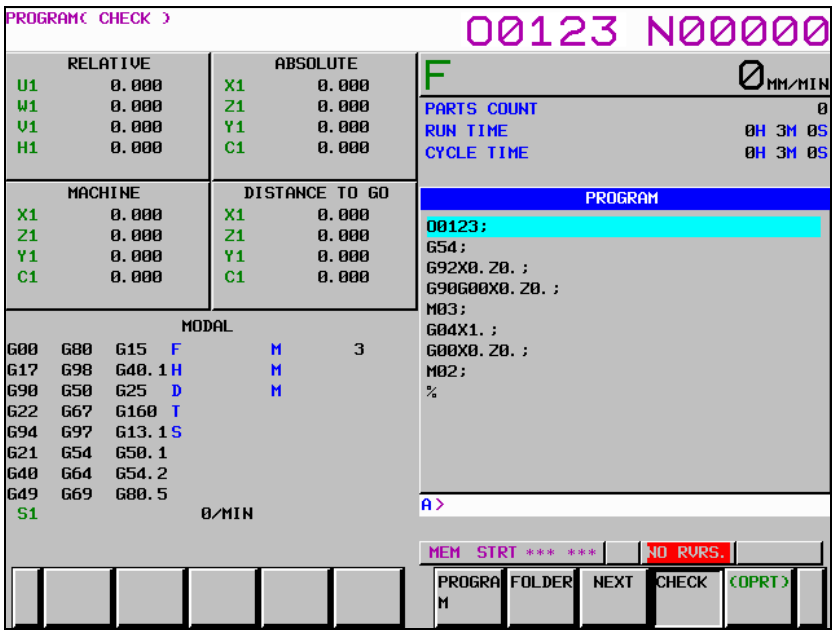


Fig. 5.8 (c) " NO RVRS." status display

Besides, when direction change prohibition signal MNCHG is set to "1" and the direction of program's execution is changed by manual handle, this status display changes from "M.H.RTR." to "NO.CHAG.". This status is displayed by blinking/reversing in the color of color number 3 (INPUT KEY, O/N NO. and STATUS are the same color) . The screen display is shown as Fig. 5.8 (d). When the program is executed in the direction as the same as before by manual handle or direction change prohibition signal MNCHG is set to "0", the " M.H.RTR." is displayed again.
Moreover, when bit 1 (FWD) of parameter No. 6400 is set to "1" and the program is executed to change direction by manual handle, this status display changes from "M.H.RTR." to "NO.CHAG.".

NOTE
When the improvement of direction change movement in auxiliary function output block is enabled, the state of direction change prohibition signal MNCHG is set to "1" and direction change is possible. Therefore please note that the status display of the direction change prohibition is displayed though it is possible to direction change.

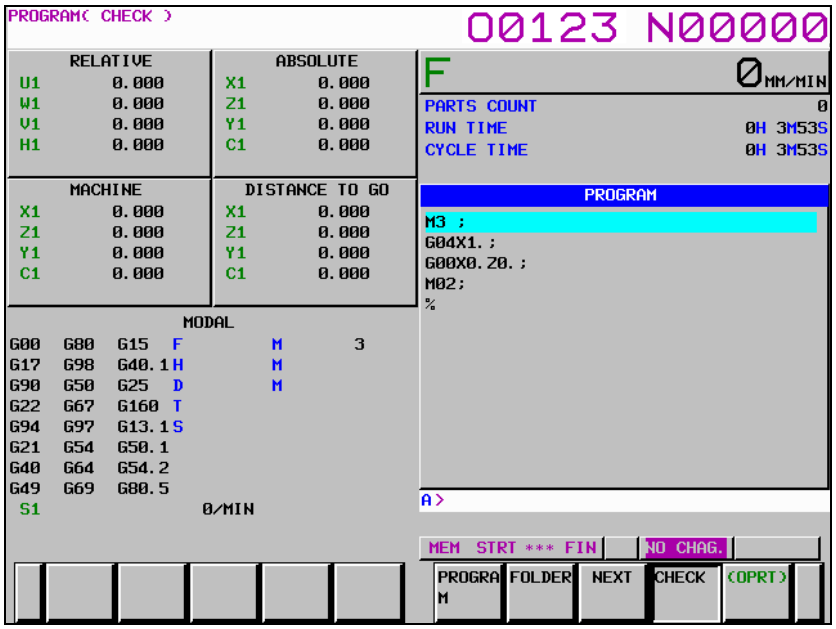


Fig. 5.8 (d) "NO.CHAG." status display

Limitation

- Movement in automatic operation by DNC operation mode(RMT)

In the automatic operation by DNC operation mode(RMT), the backward movement is prohibited though the forward movement is enable.

- Movement in subprogram operation by external subprogram call

In M198 or M-code for subprogram operation by external subprogram call (parameter No.6030), the backward movement is prohibited though the forward movement is enable.

- Movement command and M,S,T-code

When M,S,T-codes and movement commands are in the same block, the timing outputting codes changes between in forward movement and backward movement. Therefore, M, S, T-codes should be commanded in backward movement after confirming that "DEN" signal is set to "1".

Example of executing the following programs with lathe system

```
O0001 ;
M5 S0 F0 ;
G53 X0 Z0 ; ..... (1)
G1 W100 M3 S100 F1. ; ..... (2)
G0 U50. W50. ; ..... (3)
M2 ;
```

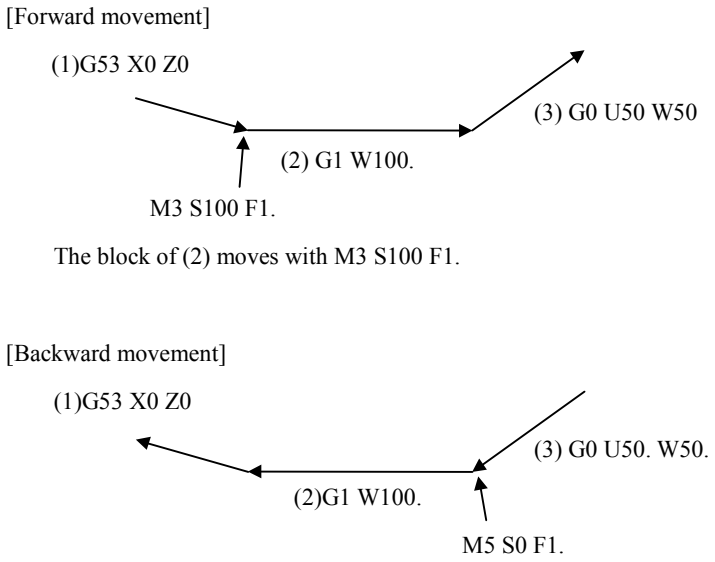


Fig. 5.8 (e)

- Non linear interpolation type positioning

In the non-linear interpolation type positioning, the route is different between forward movement and backward movement.

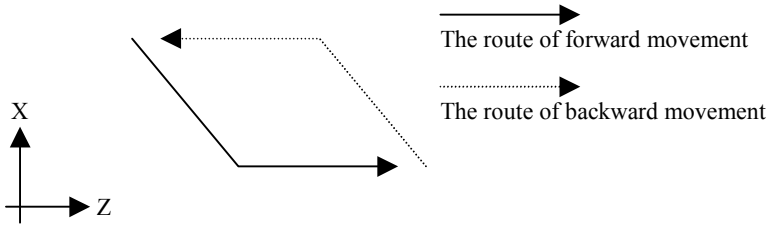


Fig. 5.8 (f)

Please use the interpolation type positioning to ward off danger. (Set the bit 1 (LRP) of parameter No.1401 to "1")
If the non-linear interpolation type positioning is used, backward movement is prohibited at the time when movement ends on any axis.

- Threading in forward movement

Threading (G32,G76,G84,G88,G92) is always executed at 100% override speed. That is to say, a pulse generated by a manual handle is ignored in executing a threading block. In thread cutting cycle, the pulse is invalid at the time actually cutting thread, but the one are valid in the other movements.

- Macro variable

In macro statement, the setting, operation, and so on of the macro variable is executed in only first forward movement. That is to say, the setting, operation, and so on of the macro variable is never executed in the block executing them once.

- Axis control by PMC

The movement of axis control by PMC cannot be controlled by this function.

- Multiple path simultaneous check in the multi-path system

When using the manual handle retrace function at the same time in multiple paths, the timing of block operation may slightly differ between these paths due to the repetition of forward and backward motion or differences in the rotation speed of the manual handle. To synchronize block operation between the paths, use the wait M code.

- Time waiting in multi-path systems

In a multi-path system, control is performed so that during forward movement, the sum of the handle pulses input from the start to the end of each block is stored and during backward movement, the tool does not move to the previous block until as many handle pulses as those stored during forward movement are input. Any handle pulses that are input in an in-position check are also stored as part of the pulses mentioned above. If, therefore, the rotation speed of the handle (that is, the axis movement speed) during forward movement differs from that during backward movement, the time required for the in-position check varies, causing a difference between the sum of the handle pulses stored during forward movement and the sum of the handle pulses during backward movement. If this occurs, even if the block ends during movement (the distance-to-go display is 0), the tool may not move to the previous block until the handle is rotated in the backward movement direction for as many handle pulses as those stored during forward movement. Moreover, in such a case, a direction change is prohibited and, therefore, it is not possible to switch to forward movement until the tool moves to the previous block by continuing backward movement.

- Check of path unit by multi-path system

In the multi-path system, the program check of an arbitrary path is possible. In the path not to check, please select the mode excluding the MEM mode. Even if bit 4 (HMP) of parameter No.6400 is set to "1", it is possible to execute the check of forward movement, direction change and backward movement.
If waiting M code exist in the program, please set No-wait signal of two path NOWT, or No-wait signal of three path NMWT to "1".

- Modal display

In the backward movement with manual handle, the modal display is updated according to the operation condition of the program.

- Modal information

In the backward movement with manual handle, the state of modal information is updated according to the operation condition of the program.

- Change in operation mode

When you change to EDIT mode during the checking mode, the backward movement and the re-forward movement cannot be executed in the blocks which have been already executed.

- ON/OFF of Manual Handle Retrace mode

When check mode signal MMOD is set to "0" and handle available signal in checking mode MCHK is set to "0", the check mode might not be turned off at once.
Basically, in the middle of block, the check mode doesn't switch from ON to OFF or from OFF to ON. After the block is ended, the check mode switches from ON to OFF or from OFF to ON.

- AI contour control

If the checking mode signal MMOD is "1", AI contour control mode (G05.1Q1) is canceled. If AI contour control is canceled, forward movement and backward movement can be executed.
Furthermore, if the signal MMOD is set to "1" in AI contour control mode, the checking mode is not effective until AI contour control mode is invalid(G05.1Q0).

- Multi Spindle

During the backward movement, both TYPE-A and TYPE-B multi spindle control may not be operated exactly.

- Path Table Operation

In path table operation, the backward movement is prohibited. Furthermore, in forward movement, regardless of manual handle, the motion is performed as the command of path table operation.

- Execution of measurement G-code with the speed of override 100%

When bit 6 (MGO) of parameter No.6400 is set to "1", a handle pulse is invalid and it is always executed at 100% override. When bit 6 (MGO) of parameter No.6400 is set to "0", this function is invalid and a handle pulse is valid.
In the multi-path system, this function is not effective in the execution of another path and handle pulse is valid in another path. The measurement G code to which this function is effective is as follows.

1) G31 for skip
2) G31, G31P1, G31P2, G31P3, G31P4, G04, G04Q1, G04Q2, G04Q3 and G04Q4 for multistage skip
3) G31P99 and G31P98 for torque limit skip

When the measurement by G36 and G37 (G37.1 and G37.2 when bit 3(G36) of parameter No.3405 is set to "1") of automatic tool compensation is executed, a handle pulse is invalid and it is always executed at 100% override regardless of setting bit 6 (MGO) of parameter No.6400. At the rapid traverse before measurement, handle pulse is valid.
When bit 7 (SKF) of parameter No.6200 is set to "0" and bit 2 (SFN) of parameter No.6207 is set to "0", handle pulse at G31 is invalid and it is always executed at 100% override regardless of setting bit 6 (MGO) of parameter No.6400.

When bit 7 (MG4) of parameter No.6400 is set to "1" and the software option of multistage skip is enabled and the setting of bits 0 (IS1) of parameter No. 6202 to bit 7 (DS8) of parameter No. 6206 is enabled, the backward movement prohibition is enabled in G04 block for multistage skip.
The G code to which this function is effective is as follows.

1) G04, G04Q1, G04Q2, G04Q03 and G04Q4 for multistage skip

- Relation to another function

This function cannot coexist with the following functions

- Retrace
- Learning control for parts cutting
- Preview repetitive control
- Learning control

5.8.1 Displaying Program Being Executed in Backward Movement

Overview

During executing the program backward by manual handle retrace, the block just before the block being executed can be displayed.
Therefore, it is possible to check what the block to be executed next is in backward movement.

Explanation

In displaying program being executed in backward movement, it is possible to select whether to display the block being executed or the block just before the block being executed at the start of the program.

Selecting method is as follows.

1 To display the block being executed at the start of the program
Set bit 2 (RPD) of parameter No.11370 to 0.

2 To display the block just before the block being executed at the start of the program
Set bit 2 (RPD) of parameter No.11370 to 1.
Furthermore, in a display containing look-ahead blocks (bit 1 (APD) of parameter No.11350 to 0), set bit 3 (FPD) of parameter No.11308 to 1.

When setting the case 2, the block just before the block selected by the cursor can be displayed.

- For a display containing look-ahead blocks (bit 1 (APD) of parameter No.11350 = 0)

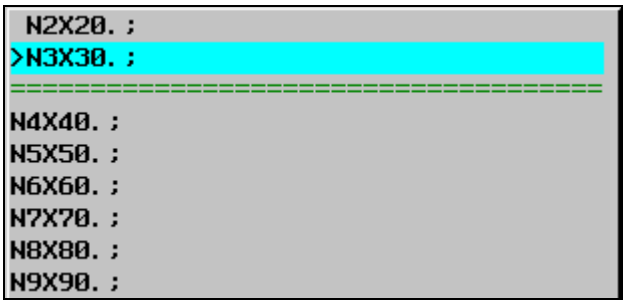


Fig. 5.8 (g) Displaying program being executed in backward movement

- For a text display (bit 1 (APD) of parameter No.11350 = 1)



Fig. 5.8 (h) Displaying program being executed in backward movement

5.9 AUXILIARY FUNCTION OUTPUT BLOCK REVERSE MOVEMENT FOR MANUAL HANDLE RETRACE

Overview

This function enables reverse movement during manual handle retrace even if a move command and an auxiliary function (M/S/T/B code) are specified in the same block.
This is an optional function.

Notes

NOTE

1 When a single auxiliary function is specified individually, carry out the regular auxiliary function completion sequence. Reverse movement becomes possible after moving to the next (or previous) block.
2 If a move command and an auxiliary function are specified in the same block and if no travel distance remains after movement, carry out the regular auxiliary function completion sequence. Reverse movement becomes possible after moving to the next (or previous) block.
3 To enable reverse movement using this function when an M code other than grouped M codes and a move command are specified in the same block, set bit 5 (RVN) of parameter No. 6400 to "0".
4 This function is not supported for M02 or M30.
5 If an auxiliary function is specified in a block having no move command, such as G70, carry out the regular auxiliary function completion sequence.
Reverse movement becomes possible after moving to the next (or previous) block.
6 This function is optional.

5.10 MANUAL HANDLE RETRACE FUNCTION FOR MULTI-PATH

Overview

In the manual handle retrace function for multi-path systems, when the operation of re-forward movement is performed, the movement timing of all-path movement can be made identical to that of forward movement.
This function is an optional function.

Explanation

In the manual handle retrace function for multi-path systems, when a certain path stopped with the backward movement prohibited and the other paths continue the backward movement, if re-forward movement is performed after that, all paths start the re-forward movement immediately, so the timing of block movement of each path is different from the original forward movement.
In this function, during re-forward movement, the path that stopped with backward movement prohibited doesn’t start re-forward movement immediately. After other paths reach the position the path stopped with backward movement prohibited, the re-forward movement of this path is performed. Therefore, in multi-path systems, it is possible to perform re-forward movement in which the timing of block movement of each path is made identical to that of forward movement.

NOTE
Even when this function is enabled, the timing of block movement of each path may differ slightly due to the repetition of forward and backward movement and the rotation speed of the manual handle. Therefore, when synchronization is necessary, use the waiting M code.

Example) Re-forward movement of manual handle retrace in a 4-path system

Path1	Path2	Path3	Path4
O1001	O1001	O1001	O1001
N1 G1 X5.0 F1	N1 G1 Z6.0 F1	N1 G1 X7.0 F1	N1 G1 Z5.0 F1
N2 X12.0	N2 G31 Z14.0	N2 X10.0	N2 G31 Z11.0
N3 X22.0	N3 Z28.0	N3 G31 X27.0	N3 Z18.0
N4 X30.0	N4 Z36.0	N4 X33.0	N4 Z29.0
N5 X34.5	N5 Z50.0	N5 X50.0	N5 Z32.0
N6 X50.0	M30	M30	N6 Z50.0
M30			M30

When the above programs are operated in each path, the operation states of individual steps (in forward, backward, and re-forward movement) are shown below.

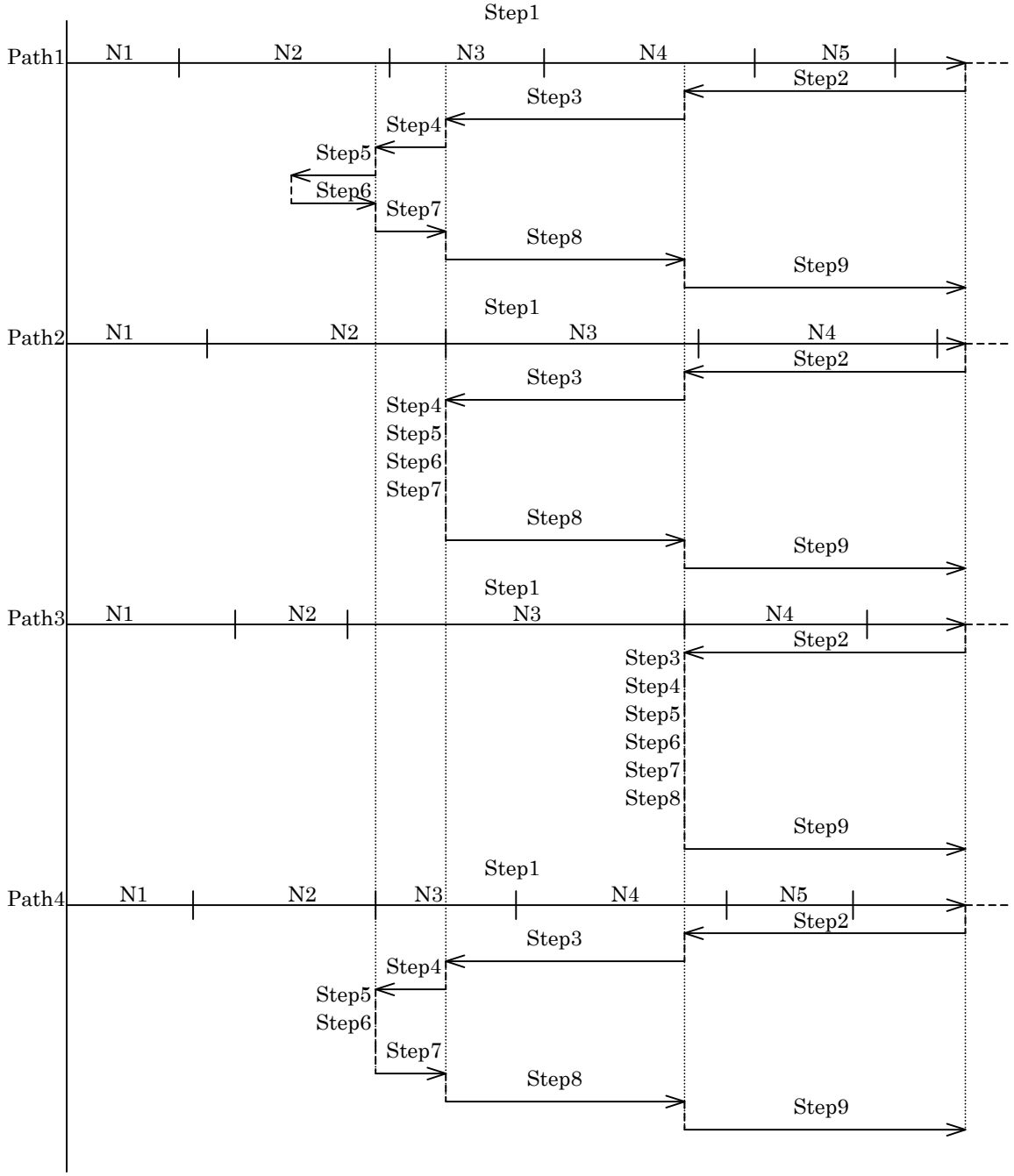


Fig. 5.10 (a)

STEP	1	2	3	4	5	6	7	8	9
Command state	Forward	Backward	Backward	Backward	Backward	Re-forward	Re-forward	Re-forward	Re-forward
Execution state in Path1	Forward	Backward	Backward	Backward	Backward	Re-forward	Re-forward	Re-forward	Re-forward
Execution state in Path2	Forward	Backward	Backward	Stop	Stop	Wait P1	Wait P1,4	Re-forward	Re-forward
Execution state in Path3	Forward	Backward	Stop	Stop	Stop	Wait P1	Wait P1,4	Wait P1,2,4	Re-forward
Execution state in Path4	Forward	Backward	Backward	Backward	Stop	Wait P1	Re-forward	Re-forward	Re-forward

However, when bit 4 (HMP) of parameter No. 6400 is 0 (even if a path is prohibited from changing movement direction, the other paths can still change it), if operation is performed in the condition shown in the example below, re-forward movement may not keep the same motion as in forward movement.

Example)

Conditions of the programs of the individual paths

Path1 In backward movement, there is a block that prohibits a change to forward movement.

Path2 In backward movement, there is neither block that prohibits backward movement nor block that prohibits a change to forward movement.

Path3 In backward movement, there is a block that prohibits backward movement.

In the above conditions, the following operation is performed.

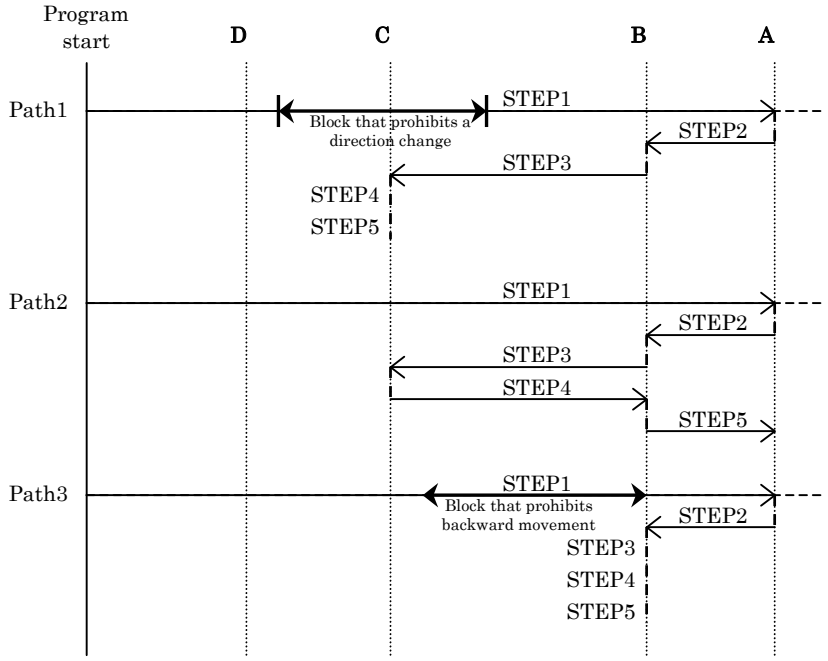


Fig. 5.10 (b)

STEP1: Forward movement is performed to position A in the figure.

STEP2, STEP3: Backward movement is performed to position C in the figure. Path 3 is prohibited from moving backward at position B.

STEP4, STEP5: Re-forward movement is performed to position A in the figure. Since path 1 is prohibited from changing its direction, only path 2 performs re-forward movement. Even when path 2 performs re-forward movement to position B, path 3 does not start re-forward movement.

To keep synchronization in re-forward movement, perform backward movement of path 1 until path 1 passes by block D, which prohibits a direction change, and start re-forward movement of path 1.
In this case, if bit 4 (HMP) of parameter No.6400 is set to 1 (if a path is prohibited from changing its movement direction, the other paths cannot also change their direction), the other paths cannot change their direction until path 1 passes by the position D.

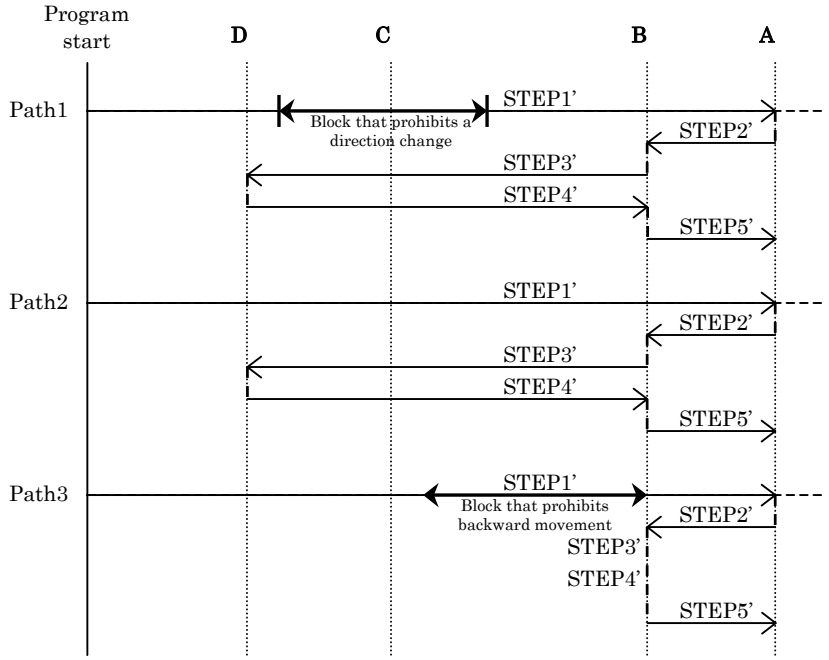


Fig. 5.10 (c)

STEP1': Forward movement is performed to position A in the figure.
STEP2',STEP3': Backward movement is performed to position D in the figure. Path 3 is prohibited from moving backward at position B.
STEP4',STEP5: Re-forward movement is performed to position A in the figure. When path 1 and path 2 perform re-forward movement to position B, path 3 starts re-forward movement.

5.11 EXTENSION OF THE MANUAL HANDLE RETRACE FUNCTION

Overview

In manual handle retrace function, the following operations that were impossible so far become available.

(1) Forward movement of rigid tapping
(2) Forward movement of threading
(3) Forward movement of PMC axis control
(4) Backward movement of the orientation by the G00 command on the Cs contour control axis
(5) Backward movement of polygon turning with two spindles
(6) Backward movement of balance cutting

This function is optional (included in the multi-path manual handle retrace function).

Explanation

- Rigid tapping

- When the bit 0 (HRA) of parameter No.6403 is set to 0 (Conventional specification)
  (1) When rigid tapping starts during forward movement, handle operation is disabled and rigid tapping is executed at a speed of 100% override.
  (2) Backward movement cannot be performed in the rigid tapping mode. In addition, when the rigid tapping block is reached after rigid tapping is completed in forward movement, backward movement cannot be done any more.
- When the bit 0 (HRA) of parameter No.6403 is set to 1 (Extensional specification)
  (1) In tapping during forward movement, the operation is performed at a speed according to handle operation.

(2) Backward movement in rigid tapping is the same movement as in the case where bit 0 (HRA) of parameter No. 6403 is 0.

- Thread cutting

- When the bit 0 (HRA) of parameter No.6403 is set to 0 (Conventional specification)
  (1) When the threading block is reached during forward movement, handle operation become disabled and the operation is performed at a speed of 100% override.
  (2) Backward movement cannot be performed during threading. In addition, when the threading block is reached during backward movement after threading is completed in forward movement, backward movement cannot be done any more.
- When the bit 0 (HRA) of parameter No.6403 is set to 1 (Extensional specification)
  (1) In threading during forward movement, the operation is performed at a speed according to handle operation.
  (2) Backward movement in threading is the same movement as in the case where bit 0 (HRA) of parameter No. 6403 is 0.

⚠ WARNING
In rigid tapping or threading during manual handle retrace, the workpiece cannot be actually machined.

- PMC axis control

- When the bit 1 (HRB) of parameter No.6403 is set to 0 (Conventional specification)
  (1) The PMC axis operates independently of handle operation.
- When the bit 1 (HRB) of parameter No.6403 is set to 1 (Extensional specification)
  (1) In PMC axis control during forward movement, the PMC axis is operated at a speed according to handle operation but, backward movement cannot be performed.

NOTE
1 Even when the axis specified by the program moves backward, the axis specified by PMC axis control cannot move backward. Therefore, the relative position between the axis specified by the program and the axis specified by PMC axis control changes when backward movement is performed while the axis that is specified by PMC axis control is operating. In this case, it is necessary to add the following PMC ladder process.
   - When the axis specified by PMC axis control is operating, prohibit the backward movement of the axis specified by the program using reverse movement prohibition signal MRVM.

2 Alarm PS0130, “NC AND PMC AXIS ARE CONFLICTED” occurs when one of the following operations is executed.
   (1) In the backward movement, when PMC axis control is specified for the axis specified by the program in the previous block.
   (2) During PMC axis control, when a direction change is performed for the axis specified by the program in the previous block.

3 In the skip (03h), continuous feed (06h), speed command (10h), torque control (11h), and auxiliary function (12h,13h,14h) of PMC axis control, even if bit 1 (HRB) of parameter No. 6403 is set to 1, the axis operates independently of handle operation.

> ⚠ **WARNING**
> During the manual handle retrace, if a reset is made when a command by PMC axis control is not completed, the command by the program stops, but the command by PMC axis control continues. In this case, even if bit 1 (HRB) of parameter No. 6403 is set to 1, the command by PMC axis control operates independently of handle operation. Make a reset after confirming that the command by the PMC axis control is completed.

- Orientation of Cs contour control axis

- When the bit 2 (HRC) of parameter No.6403 is set to 0 (Conventional specification)
  (1) Backward movement cannot be performed in orientation movement by the first G00 command after the Cs contour control mode is turned on. Therefore, when the above G00 command is reached during backward movement, backward movement cannot be done any more.
  (2) Backward movement can be performed in the reference point return command (G28) of the Cs contour control axis, but the processing proceeds to the previous block without performing orientation movement.
- When the bit 2 (HRC) of parameter No.6403 is set to 1 (Extensional specification)
  (1) Backward movement can be performed in the orientation operation by the G00 command, but the processing proceeds to the previous block without performing orientation movement as in G28 command.
  (2) The reference point return command (G28) of the Cs contour control axis performs the same movement as in the case where bit 2 (HRC) of parameter No. 6403 is 0.

> ⚠ **WARNING**
> The workpiece cannot be actually machined because the orientation movement of G00 and G28 is not executed during backward movement in Cs contour control.

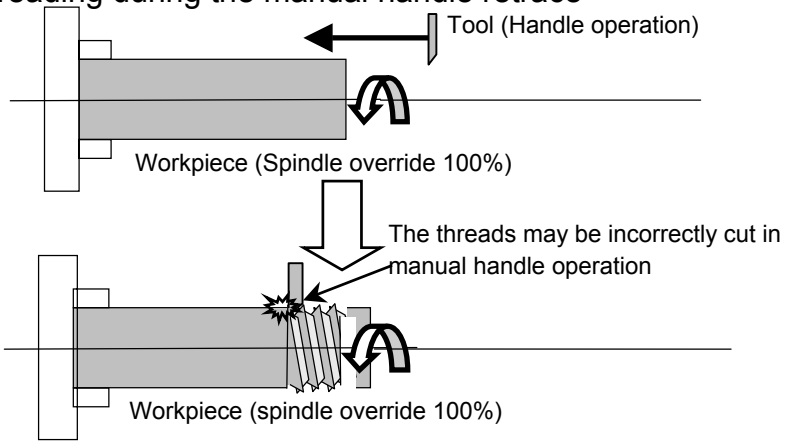
- Polygon machining with two spindles

- When the bit 3 (HRD) of parameter No.6403 is set to 0 (Conventional specification)
  (1) Axes other than the polygonal synchronization axes operate at the speed according to handle operation even in the polygon machining mode. However, backward move cannot be performed in the block specified in the polygonal machining mode.
  (2) When the polygon machining block (G50.2) is reached in backward movement after the polygon machining is completed in forward movement, backward movement cannot be done any more.
- When the bit 3 (HRD) of parameter No.6403 is set to 1 (Extensional specification)
  (1) It is possible to move backward or re-forward in the block specified in the polygonal machining mode. When backward movement is performed during polygon machining, however, if the polygon machining-on block (G51.2) is reached, backward movement cannot be done any more.
  (2) The operation when the polygon machining block (G50.2) is reached in backward movement after the polygon machining is completed in forward movement the same as in the case where bit 3 (HRD) of parameter No. 6403 is 0.

⚠ WARNING

In the threading and polygon machining between two spindles, the spindle operates at a speed of override 100% instead of the speed according to handle operation. Therefore, the workpiece cannot be actually machined.

(Example) Treading during the manual handle retrace

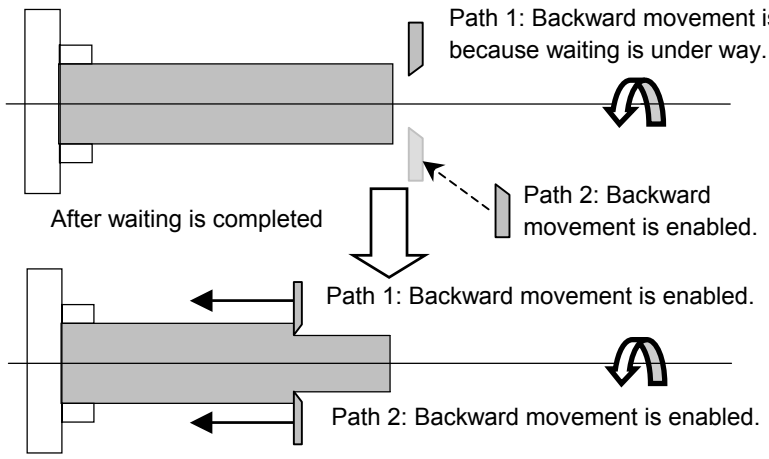


- Balanced cutting

- When the bit 4 (HRE) of parameter No.6403 is set to 0 (Conventional specification)
  (1) In the balance cutting mode, axes operate at the speed according to handle operation, but backward movement cannot be performed.
  (2) When the balance cutting block (G69) is reached in backward movement after the balance cutting is completed in forward movement, backward movement cannot be done any more.
- When the bit 4 (HRE) of parameter No.6403 is set to 1 (Extensional specification)
  (1) It is possible to move backward or re-forward in the block specified in the balance cutting mode. When backward movement is performed in the balance cutting mode, if the balance cut-on block (G68) is reached, backward movement cannot be done any more.
  (2) The operation when the balance cutting block (G69) is reached in the backward movement after the balance cutting was completed in forward movement is the same operation as in the case where bit 4 (HRE) of parameter No. 6403 is 0.

NOTE

1 In the balance cutting mode, waiting is done so that the block starts at the same time in each path. Backward movement cannot start during the waiting. After the waiting is completed and the next block starts, backward movement becomes possible.



2 In all paths that uses balance cutting, set bit 4 (HRE) of parameter No. 6403 to 1.

6 SAFETY FUNCTIONS

To immediately stop the machine for safety, press the Emergency stop button. To prevent the tool from exceeding the stroke ends, Overtravel check and Stored stroke check are available. This chapter describes emergency stop, overtravel check, and stored stroke check.

Chapter 6, "SAFETY FUNCTIONS", consists of the following sections:

6.1 EMERGENCY STOP 1134
6.2 OVERTRAVEL 1134
6.3 STORED STROKE CHECK 1135
6.4 STROKE LIMIT CHECK BEFORE MOVE 1139
6.5 STROKE LIMIT AREA CHANGING FUNCTION 1141
6.6 WRONG OPERATION PREVENTION FUNCTIONS 1142

6.1 EMERGENCY STOP

If you press Emergency Stop button on the machine operator's panel, the machine movement stops in a moment.

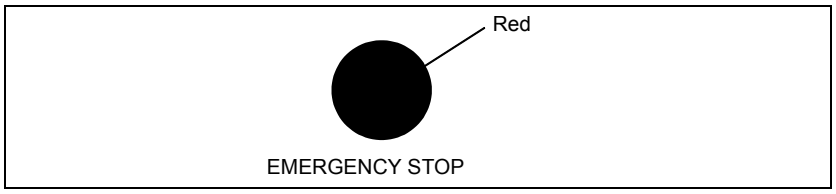


Fig. 6.1 (a) Emergency stop

This button is locked when it is pressed. Although it varies with the machine tool builder, the button can usually be unlocked by twisting it.

Explanation

EMERGENCY STOP interrupts the current to the motor.
Causes of trouble must be removed before the button is released.

6.2 OVERTRAVEL

When the tool reaches the stroke end set by the machine tool limit switch, it decelerates and stops because of working the limit switch and an OVERTRAVEL alarm is displayed.

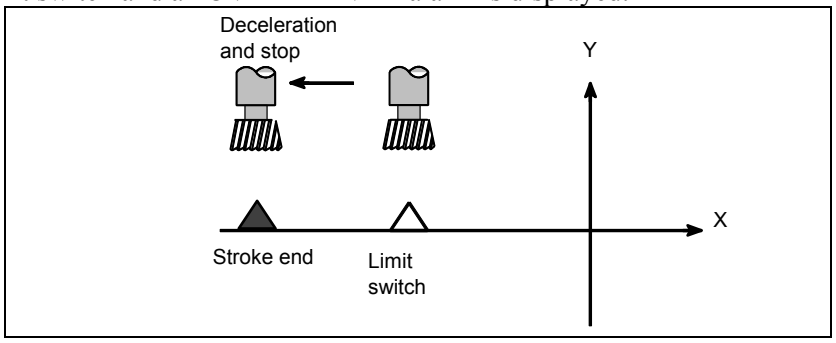


Fig. 6.2 (a) Overtravel

Explanation

- Overtravel during automatic operation

When the tool touches a limit switch along an axis during automatic operation, the tool is decelerated and stopped along all axes and an overtravel alarm is displayed.

- Overtravel during manual operation

In manual operation, the tool is decelerated and stopped only along the axis for which the tool has touched a limit switch. The tool still moves along the other axes.

- Releasing overtravel

Press the reset button to reset the alarm after moving the tool to the safety direction by manual operation. For details on operation, refer to the operator's manual of the machine tool builder.

Alarm

Table6.2 (a)

Alarm No.	Message	Description
OT0506	+ OVERTRAVEL (HARD)	The stroke limit switch in the positive direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is not generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.
OT0507	- OVERTRAVEL (HARD)	The stroke limit switch in the negative direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is not generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.

6.3 STORED STROKE CHECK

Three areas which the tool cannot enter can be specified with stored stroke check 1, stored stroke check 2, and stored stroke check 3.

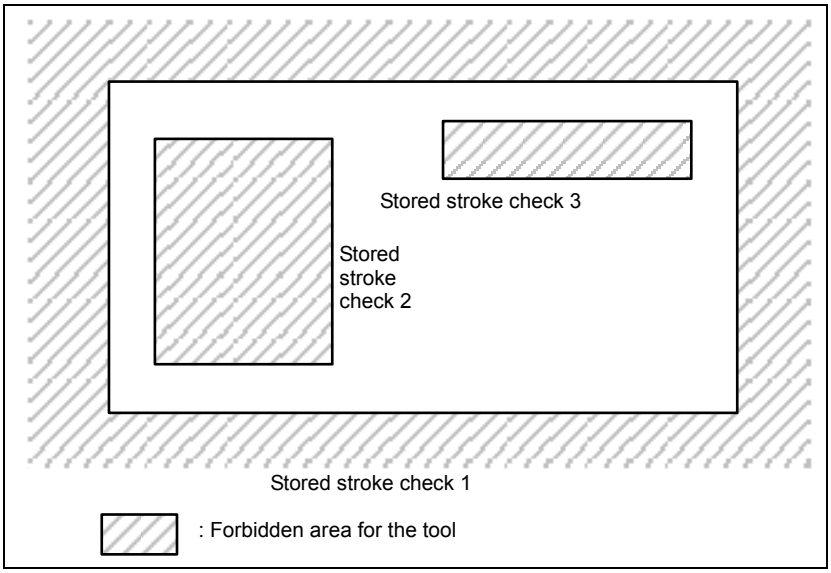


Fig. 6.3 (a) Stroke check

The following shows the areas which the tool cannot enter for each stored stroke check.

- Stored stroke check 1: Outside
- Stored stroke check 2: Outside or inside (switchable)

- Stored stroke check 3: Inside

When the tool moves into the forbidden area, an alarm is displayed and the tool is decelerated and stopped.
When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came.
The stored stroke check 2 and 3 functions are optional.

Explanation

- Stored stroke check 1

Parameters (Nos. 1320, 1321 or Nos. 1326, 1327) set boundary. Outside the area of the set limits is a forbidden area. The machine tool builder usually sets this area as the maximum stroke.
When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came.
At this time, a signal (overtravel alarm signal) can be output to the PMC if bit 6 (OTS) of parameter No. 1301 is set to 1. In addition, when the tool enters the forbidden area during manual operation, the signal (overtravel alarm signal) can be output to the PMC without generating the alarm by setting bit 1 (NAL) of parameter No. 1300 to 1. With this parameter setting, the alarm is generated when the tool enters the forbidden area during automatic operation.

⚠ CAUTION
1 If the two points for specifying a forbidden area are identical, all areas are handled as forbidden areas for stored stroke check 1.
2 The size of a forbidden area must be set carefully. If the size is set incorrectly, the stroke becomes infinite.

- Stored stroke check 1 area expansion

Stored stroke check 1 area expansion enables definition of eight areas using parameters Nos. 1350 to 1361.

NOTE
The basic notes are the same as those of stored stroke check 1.

- Stored stroke check 2

Parameters Nos. 1322, 1323 or commands set these boundaries. Inside or outside the area of the limit can be set as the forbidden area. Bit 0 (OUT) of parameter No. 1300 selects either inside or outside as the forbidden area.
In case of program command a G22 command forbids the tool to enter the forbidden area, and a G23 command permits the tool to enter the forbidden area.
Each of G22; and G23; should be commanded independently of another commands in a block.
The command in the Fig. 6.3 (b) creates or changes the forbidden area:

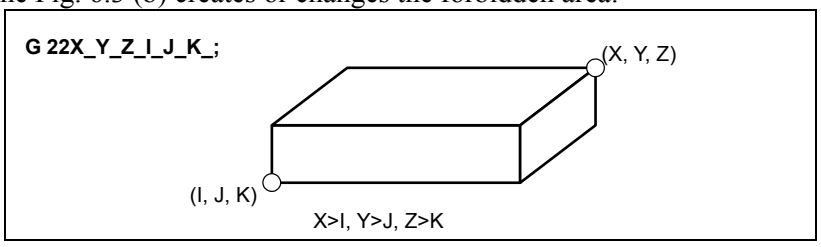


Fig. 6.3 (b) Creating or changing the forbidden area using a program

When setting the area by parameters, points A and B in the Fig. 6.3 (c) must be set.

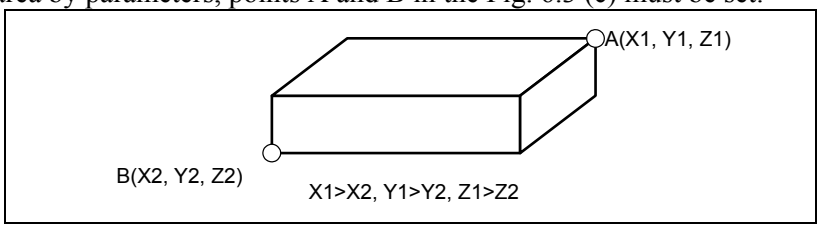


Fig. 6.3 (c) Creating or changing the forbidden area using a parameters

The values X1, Y1, Z1, X2, Y2, and Z2, which are set by parameters Nos. 1322 and 1323, must be specified by the distance from the machine coordinate system (machine unit). The values X, Y, Z, I, J, and K, which are set by a G22 command, must be specified by the distance in the least input increment (input unit).
Values set by a program are then converted in the machine increment and the values are set as the parameters.

- Stored stroke check 3

Set the boundary with parameters Nos. 1324 and 1325. The area inside the boundary becomes the forbidden area. The values X1, Y1, Z1, X2, Y2, and Z2 must be set as coordinates (machine unit) in the machine coordinate system.

⚠ CAUTION
1 If the two points for specifying a forbidden area are identical, all areas are handled as movable areas for stored stroke check 2/3.
2 Even if the two points for specifying a forbidden area are erroneously set, the rectangular parallelepiped having the points as vertices is assumed as a boundary.
3 Since an axis without the reference position return function has no forbidden areas, there are no alarms about forbidden areas for the axis.

- Checkpoints in the forbidden area

The parameter setting or programmed value (XYZIJK) depends on which part of the tool or tool holder is checked for entering the forbidden area.
If point A (the top of the tool) is checked in Fig. 6.3 (e), the distance "a" should be set as the data for the stored stroke limit function. If point B (the tool chuck) is checked, the distance "b" must be set. When checking the tool tip (like point A), and if the tool length varies for each tool, setting the forbidden area for the longest tool requires no re-setting and results in safe operation.

- For machining center system

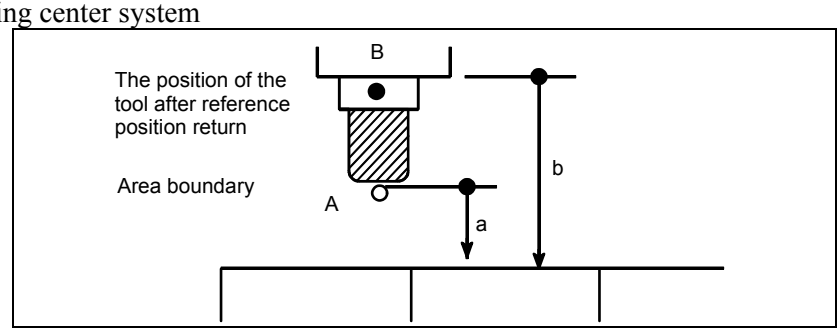


Fig. 6.3 (d) Setting the forbidden area

- For lathe system

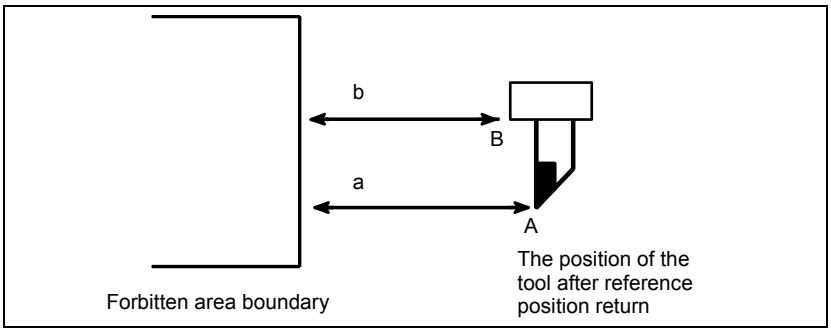


Fig. 6.3 (e) Setting the forbidden area

- Forbidden area overlapping

Area can be set in piles.

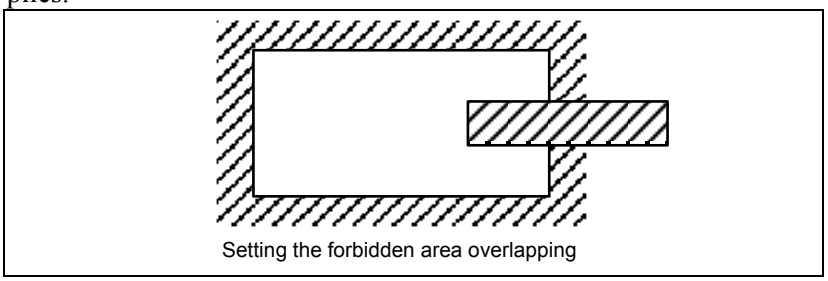


Fig. 6.3 (f) Setting the forbidden area overlapping

Unnecessary limits should be set beyond the machine stroke.

- Condition under which each check is enabled

Each check becomes effective after the power is turned on and manual reference position return or automatic reference position return by G28 has been performed.
After the power is turned on, if the reference position is in the forbidden area of each limit, an alarm is generated immediately. (Only in G22 mode for stored stroke check 2).

- Releasing the alarms

If the enters a forbidden area and an alarm is generated, the tool can be moved only in the backward direction. To cancel the alarm, move the tool backward until it is outside the forbidden area and reset the system. When the alarm is canceled, the tool can be moved both backward and forward.

- Change from G23 to G22 in a forbidden area

When G23 is switched to G22 in the forbidden area, the following results.
<1> When the forbidden area is inside, an alarm is informed in the next move.
<2> When the forbidden area is outside, an alarm is informed immediately.

- Timing for displaying an alarm

In stored stroke check 1/2/3, parameter BFA (bit 7 of No. 1300) selects whether an alarm is displayed immediately before the tool enters the forbidden area or immediately after the tool has entered the forbidden area.

Alarm

Number	Message	Description
OT0500	+ OVERTRAVEL (SOFT 1)	A movement in the positive direction exceeded stored stroke check 1.
OT0501	- OVERTRAVEL (SOFT 1)	A movement in the negative direction exceeded stored stroke check 1.
OT0502	+ OVERTRAVEL (SOFT 2)	A movement in the positive direction exceeded stored stroke check 2.
OT0503	- OVERTRAVEL (SOFT 2)	A movement in the negative direction exceeded stored stroke check 2.

Number	Message	Description
OT0504	+ OVERTRAVEL (SOFT 3)	A movement in the positive direction exceeded stored stroke check 3.
OT0505	- OVERTRAVEL (SOFT 3)	A movement in the negative direction exceeded stored stroke check 3.

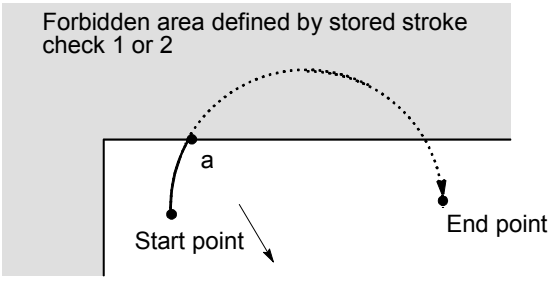
6.4 STROKE LIMIT CHECK BEFORE MOVE

During automatic operation, before the movement specified by a given block is started, whether the tool enters the forbidden area defined by stored stroke check 1, 2, or 3 is checked by determining the position of the end point from the current position of the machine and a specified amount of travel. If the tool is found to enter the forbidden area defined by a stored stroke limit, the tool is stopped immediately upon the start of movement for that block, and an alarm is displayed.

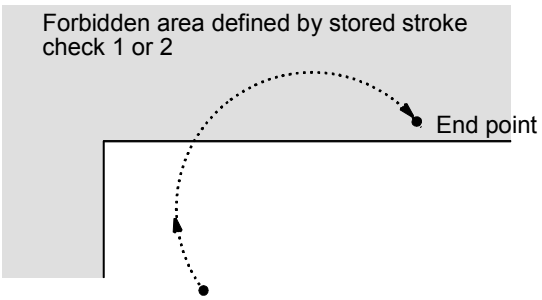
⚠ WARNING

Whether the coordinates of the end point, reached as a result of traversing the distance specified in each block, are in a forbidden area is checked. In this case, the path followed by a move command is not checked. However, if the tool enters the forbidden area defined by stored stroke check 1, 2, or 3, an alarm is issued. (See the examples below.)

Example 1)

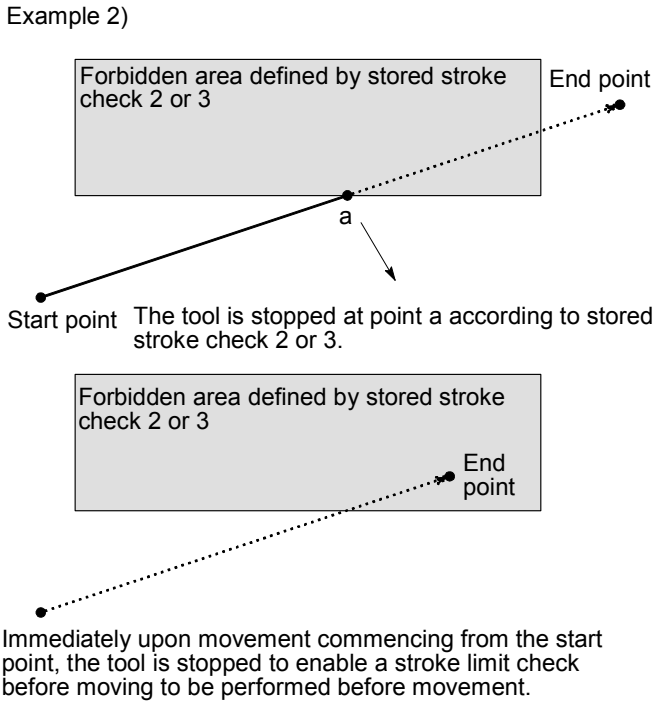


The tool is stopped at point a according to stored stroke check 1 or 2.



Immediately upon movement commencing from the start point, the tool is stopped to enable a stroke limit check before moving to be performed before movement.

⚠ WARNING



Explanation

When a stroke limit check before moving is performed, whether to check the movement performed by a G31 (skip) block and G37 (automatic tool length measurement) block can be determined using (bit 2 (NPC) of parameter No. 1301).

Limitation

- Machine lock

If machine lock is applied at the start of movement, no stroke limit check made before movement is performed.

- G23

When stored stroke check 2 is disabled (G23 mode), no check is made to determine whether the tool enters the forbidden area defined by stored stroke check 2.

- Program restart

When a program is restarted, an alarm is issued if the restart position is within a forbidden area.

- Manual intervention following a feed hold stop

When the execution of a block is restarted after manual intervention following a feed hold stop, no alarm is issued even if the end point following a manual intervention is within a forbidden area.

- A block consisting of multiple operations

If a block consisting of multiple operations (such as a canned cycle and exponential interpolation) is executed, an alarm is issued at the start point of any operation whose end point falls within an inhibited area.

- Cylindrical interpolation mode

In cylindrical interpolation mode, no check is made.

- Polar coordinate interpolation mode

In polar coordinate interpolation mode, no check is made.

- 3-dimensional coordinate system conversion

In 3-dimensional coordinate system conversion mode, no check is made.

- PMC axis control

No check is made for a movement based on PMC axis control.

Alarm

Table 6.4 (a) Alarm

Number	Message	Description
OT0510	+ OVERTRAVEL (PRE-CHECK)	The block end point was found in the + side stroke limit prohibition area during a stroke check before movement. Modify the program.
OT0511	- OVERTRAVEL (PRE-CHECK)	The block end point was found in the - side stroke limit prohibition area during a stroke check before movement. Modify the program.

6.5 STROKE LIMIT AREA CHANGING FUNCTION

Overview

This function can be used to rewrite the parameters that set the + side coordinate value and the - side coordinate values of the stroke limits even when the axis is traveling. The new stroke limit range is enabled immediately after the parameters are rewritten. The rewritable parameters are shown below.

- Parameter No. 1320
  Coordinate value I of stored stroke check 1 in the positive direction on each axis
- Parameter No. 1321
  Coordinate value I of stored stroke check 1 in the negative direction on each axis
- Parameter No. 1322
  Coordinate value of stored stroke check 2 in the positive direction on each axis
- Parameter No. 1323
  Coordinate value of stored stroke check 2 in the negative direction on each axis
- Parameter No. 1324
  Coordinate value of stored stroke check 3 in the positive direction on each axis
- Parameter No. 1325
  Coordinate value of stored stroke check 3 in the negative direction on each axis
- Parameter No. 1326
  Coordinate value II of stored stroke check 1 in the positive direction on each axis
- Parameter No. 1327
  Coordinate value II of stored stroke check 1 in the negative direction on each axis

The machining cycle time can be reduced because this function can rewrite parameters even if some axes are moving. This function is optional.

Limitation

- Axis synchronous control

This function is disabled when the automatic slave axis parameter setting is enabled (bit 4 (SYP) of parameter No. 8303 = 1) in feed axis synchronous control. (Writing to the parameters is allowed only when all axes are stopped.)

Note

NOTE

If the parameters are rewritten so that the current position is included in a forbidden area during axis movement, the axis decelerates and stops, and an alarm is displayed. If an alarm occurs when the tool enters a forbidden area, the tool can move only opposite to the direction from which the tool came. After moving it outside the forbidden area in the opposite direction, release the alarm with a reset. When the alarm is released, the tool can move in both directions.

6.6 WRONG OPERATION PREVENTION FUNCTIONS

An improper tool offset setting or an improper operation of the machine can result in the workpiece being cut inadequately or the tool being damaged. Also, if data is lost due to an operation mistake, it takes extra time to recover from the mistake.
The wrong operation prevention functions described below are meant to prevent the operator from performing any unintended operation (hereinafter referred to as an improper operation).

1 Functions that are used when data is set
- Data check to verify that the offset data is within the valid setting range
- Incremental input operation confirmation
- Prohibition of the absolute input by the soft key to prevent any improper absolute or incremental input operation
- Confirmation of any operation of deleting the program or all data
- Confirmation of a data update during the data setting process

2 Functions that are used when the program is executed
- Highlighting of updated modal information
- Display of the executed block status prior to the program execution
- Display of the axis status, such as the mirror image function enabled or the interlock function enabled
- Check for starting from the middle of the program
- Data check to verify that the offset data is within the effective setting range
- Maximum incremental value check

6.6.1 Functions that are Used When Data is Set

The following functions are provided to prevent improper operations when data is set.
- Input data range check
- Confirmation of incremental input
- Prohibition of the absolute input by the soft key
- Confirmation of the deletion of the program
- Confirmation of the deletion of all data
- Confirmation of a data update during the data setting process

Set these functions on the operation confirmation function setting screen. For the input data range check, set a valid input data range, e.g. the upper and lower limits, for each input screen. For the other functions, specify whether to enable or disable them.
For information about how to display the individual setting screens, how to manipulate them, and other details, see the item “Operation confirmation setting screen” that describes the operation procedures.

6.6.1.1 Input data range check

This function allows an effective data range to be set and checks whether the input data is within the set range.

Input data range check

Explanation

- Outline of the input data range check

This function allows an effective data range to be set for the data of each input screen listed later and checks whether the input data is within the set range. If the input data is out of the effective data range, the warning message "DATA IS OUT OF RANGE" is displayed and the data is rejected.

For example, assume that the effective data range for a certain tool offset number is set to -200. to 200, and that you are going to input 100.[INPUT]. Even if you inadvertently press the 0 key one more time, resulting in 1000.[INPUT], the input of 1000. is not accepted.
The function detects a setting mistake and prevents the program from running with invalid data.

- Input screens for which this function is effective

- Tool compensation
- Workpiece origin offset

T

- Y-axis tool offset
- Workpiece shift

- Settings

To enable this function, set an effective data range for each input screen on the operation confirmation function setting screen. For information about how to display the individual setting screens, how to set data ranges, and other details, see the items that describe the setting of the data ranges.
If the set data range is invalid, no data input is accepted. Correct the data range setting, and then input data.

- Disabling the function

The input data range check is disabled if you make any of the following settings on the operation confirmation function setting screen.

- Both the upper and lower limit values for the tool offset number or workpiece coordinate system are 0.
- The upper and lower limit values for each offset are identical.

- Messages displayed during the input data range check

When the cursor moves into an input field of an input screen, one of the messages and warning messages in the Table 6.6.1.1 (a) or Table 6.6.1.1 (b) is displayed. No message is displayed when the input data range check is disabled.

When the set effective data range is valid

Table 6.6.1.1 (a) List of messages displayed 1

Input data status	Message	Color
The data in the input field is within the range.	Input range xxx - xxx	Black
The data in the input field is out of the range.	Input range xxx - xxx	Red

xxx: Upper and lower limit values

When the set effective data range is invalid

Table 6.6.1.1 (b) List of messages displayed 2

Range check status	Message	Color
Tool offset number overlap	NG SETTING (OFFSET NUM OVERLAP)	Red
Workpiece coordinate system overlap	NG SETTING (WORK COORD VAL OVERLAP)	Red
Invalid upper and lower limit values	NG SETTING (U-LMT AND L-LMT ILLEGAL)	Red

The message "NG SETTING (U-LMT AND L-LMT ILLEGAL)" is displayed in the following cases:

- The upper and lower limit values are reversed.
- The values are not effective (e.g., more pairs of offset numbers than allowed are set).
- Either of the tool offset numbers is 0.

- Range check for data changed by G10 or system variable

If the data changed by G10 or system variable is out of the effective data range, the alarm PS0334 "OFFSET DATA OUT OF RANGE" is displayed.

6.6.1.2 Confirmation of incremental input

This function displays a confirmation message when you attempt to input an incremental value by using the [+INPUT] soft key.

Confirmation of incremental input

Explanation

- Outline of the confirmation of incremental input

This function displays a confirmation message when you attempt to input an incremental value by using the [+INPUT] soft key in any of the input screens listed below. It lets you confirm whether you really want to change data or not before making that change.

For example, when you set 5.[+INPUT] for 10., the message "15. INPUT OK?" is displayed.
The function prevents improper absolute or incremental input operations.

NOTE
This function cannot be used to input two or more values consecutively by delimiting them by semicolons (;).

- Input screens for which this function is effective

- Tool compensation
- Workpiece origin offset
- Settings
- Parameter
- Pitch error compensation

M

- Chopping

T

- Workpiece shift
- Y-axis tool offset
- Second tool geometry offset
- Chuck tail stock barrier
- Tool geometry data

- Settings

In the operation confirmation function setting screen, check or uncheck the "INCREMENTAL INPUT" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.6.1.3 Prohibition of the absolute input by the soft key

This function prohibits the absolute input using the [INPUT] soft key.

Prohibition of the absolute input by the soft key

Explanation

- Outline of the prohibition of the absolute input by the soft key

This function prohibits the absolute input by the [INPUT] soft key in the input screens listed later.
It prevents improper absolute or incremental input operations by requiring that the absolute input be made using the 

MDI key and that the incremental input be made using the [+INPUT] soft key.

- Input screens for which this function is effective

- Tool compensation
- Workpiece origin offset

T

- Y-axis tool offset
- Workpiece shift

- Settings

In the operation confirmation function setting screen, check or uncheck the "DISABLED SOFTKEY[INPUT] IN" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.6.1.4 Confirmation of the deletion of the program

This function displays the confirmation message "DELETE PROGRAM ?" when you attempt to delete the program.

Confirmation of the deletion of the program

Explanation

- Outline of the confirmation of the deletion of the program

When you attempt to delete the program, this function displays the confirmation message "DELETE PROGRAM ?" It lets you confirm whether you really want to delete the program or not before executing the deletion.
The function prevents the program from being deleted due to an improper operation.

- Settings

In the operation confirmation function setting screen, check or uncheck the "PROGRAM DELETE" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.6.1.5 Confirmation of the deletion of all data

This function displays the confirmation message "DELETE ALL DATA?" when you attempt to delete all data.

Confirmation of the deletion of all data

Explanation

- Outline of the confirmation of the deletion of all data

When you attempt to delete all data on the input screen described later, this function displays the confirmation message "DELETE ALL DATA?". It lets you confirm whether you really want to delete all data or not before executing the deletion.
The function prevents all data from being deleted due to an improper operation.

- Input screens for which this function is effective

- Tool compensation

T

- Y-axis tool offset

- Settings

In the operation confirmation function setting screen, check or uncheck the "ALL DATA DELETE" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.6.1.6 Confirmation of a data update during the data setting process

This function displays the [CAN] and [EXEC] soft keys for confirmation when you attempt to update the data of an input screen during the data setting process.

Confirmation of a data update during the data setting process

Explanation

- Outline of the confirmation of a data update during the data setting process

When you input data in input screen during the data setting process, this function displays the [CAN] and [EXEC] soft keys for confirmation. It lets you confirm whether you really want to update the data or not before executing the update.
The function prevents set values from being lost due to an improper operation.

If you input data using the [+INPUT] soft key when the confirmation of incremental input is enabled, a message is displayed to confirm the incremental input.

- Settings

In the operation confirmation function setting screen, check or uncheck the "INPUT IN SETTING" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.6.2 Functions that are Used when the Program is Executed

Overview

The following functions are provided to prevent improper operations when the program is executed.

- Display of updated modal information
- Start check signal
- Axis status display
- Confirmation of the start from a middle block
- Data range check
- Maximum incremental value check

In the operation confirmation function setting screen, specify whether to enable or disable these functions individually.
For information about how to display the setting screen, how to manipulate it, and other details, see the item “Operation confirmation setting screen” that describes the operation procedures.

6.6.2.1 Display of updated modal information

This function allows modal information updated by the NC command or RESET to be highlighted in the modal information display for the current block.

Display of updated modal information

Explanation

- Outline of the display of updated modal information

This function allows modal information updated by the NC command or RESET to be highlighted in the modal information display for the current block.
For example, when an absolute command has been changed to an incremental command or when the workpiece coordinate system has been initialized by RESET, the function displays the changed part of the data in an easy-to-recognize manner, in order to prevent improper operations during the execution of the program.

- Settings

In the operation confirmation function setting screen, check or uncheck the "UPDATE MODAL HIGHLIGHT DISPLAY" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.6.2.2 Start check signal

This function displays the remaining amount of travel and modal information of the block to be executed and puts the program to a temporary halt before the program is executed.

Start check signal

Explanation

- Outline of the start check signal

When a cycle start is made with the start check signal STCHK <G0408.0> set to 1, the function displays the remaining amount of travel and modal information of the block to be executed and puts the program to a temporary halt. Making the cycle start again resumes the execution of the program.
The function lets you check the status of the block before executing it, thus helping to prevent improper operations at the time of execution.

Using this function in combination with the updated modal information display function described in the preceding subsection makes it easier to check the status of the block to be executed.

- Settings

This function does not require any setting on the operation confirmation function setting screen.

6.6.2.3 Axis status display

This function displays the axis status to the left of the axis name in the coordinate display screen.

Axis status display

Explanation

- Outline of the axis status display

This function displays the axis status to the left of the axis name in the display of the machine coordinates, absolute coordinates, relative coordinates, and remaining travel amounts.
For example, when the mirror image function is enabled for the X1 axis, the absolute coordinates are displayed as follows.

```
ABSOLUTE
M  X1     10.000
   Y1     10.000
   Z1      0.000
```

By displaying the axis status as shown above, the function prevents improper operations at the time of execution.

- Axis status indication

The axis status is indicated as follows. These indications are listed in order of priority.

AXIS DETACH	: D
INTERLOCK	: I
MACHINE LOCK	: L
SERVO OFF	: S
AXIS MOVING	: *
MIRROR IMAGE	: M

- Settings

In the operation confirmation function setting screen, check or uncheck the "AXIS STATUS DISPLAY" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

6.6.2.4 Confirmation of the start from a middle block

This function displays a confirmation message when you attempt to execute a memory operation with the cursor placed on a block in the middle of the program.

Confirmation of the start from a middle block

Explanation

- Outline of the confirmation of the start from a middle block

This function displays the confirmation message "START FROM MIDDLE OF PROG (START/RESET)" when you attempt to execute a memory operation with the cursor placed on a block in the middle of the program. It lets you confirm whether you really want to start execution from that block or not before executing the program.
The function prevents you from inadvertently making a cycle start from a block in the middle of the program.

- Settings

In the operation confirmation function setting screen, check or uncheck the "START FROM MIDDLE OF PROGRAM" box to enable or disable this function. For information about how to display the setting screen, how to set the function, and other details, see the item “Operation confirmation setting” that describes the setting of the operation confirmation function.

- Multi-path system

The operation of the reconfirmation of the start from a middle block in a multi-path system is determined with the setting of bit 0 (MSC) of parameter No. 10335.
For example, if a cycle start is executed on two paths simultaneously in the program below, the operation is as given in Table 6.6.2.4.

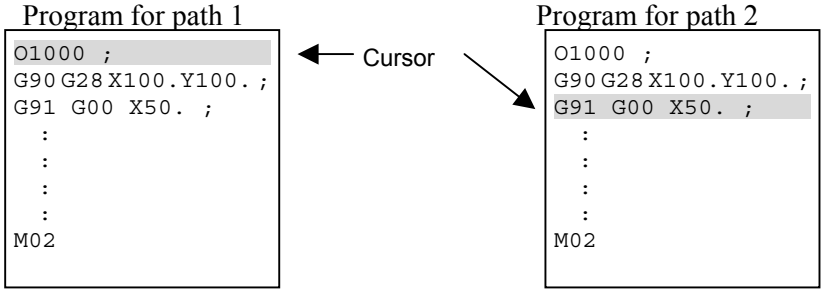


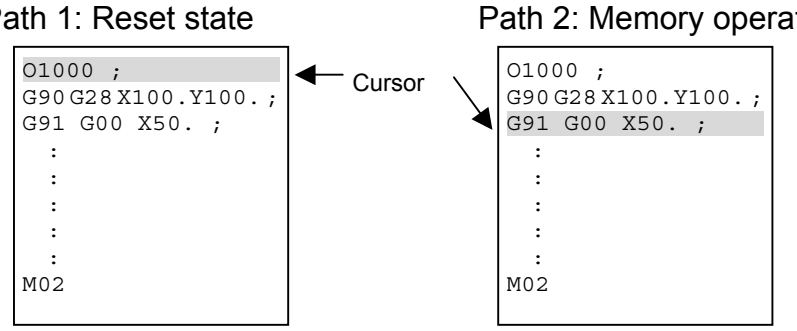
Table 6.6.2.4 Operation in a multi-path system

Bit 0 (MSC) of parameter No. 10335		Memory operation	Confirmation message
Path 1	Path 2		
0	0	Starts on path 1 only.	Displayed for both path.
0	1		
1	0		
1	1	Does not start on either path.	Displayed for both path.

To reconfirm the start from a middle block between paths, set bit 0 (MSC) of parameter No. 10335 to 1 for each path.

NOTE

If bit 0 (MSC) of parameter No. 10335 for each path is 1, a cursor position check is not performed on the path on which memory operation is in progress. For example, in the case below, if a cycle start is executed on path 1, a cursor position check is not performed on path 2.



6.6.2.5 Data range check

This function lets you set an effective data range and check whether the data to be used for execution is within the set range.

Data range check

Explanation

- Outline of the data range check

This function lets you set an effective data range for each data item listed later and check whether the data to be used for execution is within the set range. If the data is out of the effective range, the alarm PS0334 "OFFSET DATA OUT OF EFFECTIVE RANGE" is displayed.
The function detects data setting mistakes and prevents the program from running with invalid data.

- Data for which this function is effective

- Tool compensation
- Workpiece origin offset

T

- Y-axis tool offset
- Workpiece shift

NOTE

To use this function, you need to set each effective data range correctly. For information about how to set the data ranges, see the item “Effective value range for each data”.

6.6.2.6 Maximum incremental value check

This function checks the maximum incremental value specified for each axis by the NC command.

Maximum incremental value check

Explanation

- Outline of the maximum incremental value check

When the maximum incremental value is specified by the NC command described later, make sure that the absolute value of the travel distance by the incremental command does not exceed the specified value. If the specified value is exceeded, the alarm PS0337 "EXCESS MAXIMUM INCREMENTAL VALUE" is displayed.
A maximum incremental value can be specified on a per-axis basis and remains effective until 0 is set or the value is reset.
For example, when AI contour control is used, the function checks whether the amount of movement between blocks is kept to the specified value or less. Through this process, it detects erroneous program settings and prevents the program from running with invalid data.

- Format

The format of the NC command used to specify the maximum incremental value is as follows.

G91.1 IP_ ;
IP_ ; Maximum incremental value
(To cancel the maximum incremental value check, set 0.)

6.6.3 Setting Screen

This section describes how to display the operation confirmation function setting screen and how to set the individual data items on this screen.
The operation confirmation function setting screen allows you to set the following items:

- Enabling or disabling each operation confirmation function
- Effective value range for the tool offset
- Effective value range for the workpiece origin offset

T

- Effective value range for the Y-axis tool offset
- Effective value range for the work shift

6.6.3.1 Operation confirmation function setting screen

This screen displays the enable/disable setting status of the following operation confirmation functions and lets you change their settings. (Hereinafter, the screen is referred to as the operation confirmation function setting screen.)

- Confirmation of incremental input
- Prohibition of the absolute input by the soft key
- Confirmation of the deletion of the program
- Confirmation of the deletion of all data
- Confirmation of a data update during the data setting process
- Display of updated modal information
- Axis status display
- Confirmation of the start from a middle block

Displaying and setting the operation confirmation function setting screen

Procedure

1 Press the  function key.

2 Press the continuous menu key  at the right edge of the screen several times until the [GUARD] soft key is displayed.

3 Press the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).

4 If any screen other than the operation confirmation function setting screen is displayed, press the [GUARD] soft key. The operation confirmation function setting screen is displayed.

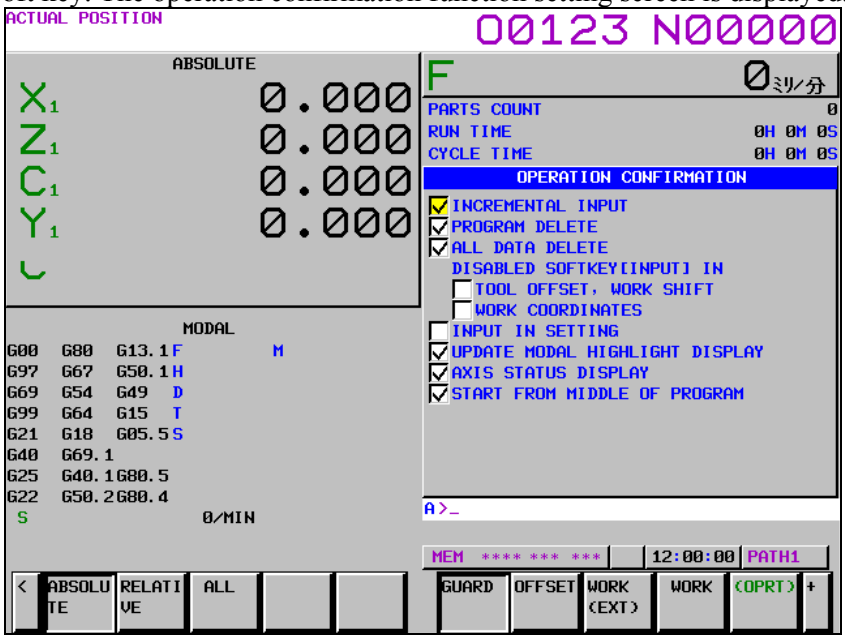


Fig. 6.6.3.1 (a) Operation confirmation function setting screen

5 Move the cursor to the check box of the item you want to change the setting, by pressing the , , , and  keys.

6 Press the soft key [(OPRT)].

7 Press the soft key [ON:1] or [OFF:0]. When you press the [ON:1] soft key, a check mark (✓) appears in the corresponding check box, indicating that the function is enabled. When you press the [OFF:0] soft key, the check mark disappears from the check box, indicating that the function is disabled.

Explanation

- Items to be set

The Table 6.6.3.1 (a) shows what is displayed for each item to be set and the corresponding functions. The items whose column of "Default" is marked by "○" is effective automatically after memory all clear.

Table 6.6.3.1 (a)

Displayed item	Default	Corresponding function
INCREMENTAL INPUT	○	Confirmation of incremental input
DISABLED SOFTKEY[INPUT] IN TOOL OFFSET, WORK SHIFT		Prohibition of the absolute input by the soft key (tool offset, Y-axis tool offset (lathe system), and work shift (lathe system))
DISABLED SOFTKEY[INPUT] IN WORK COORDINATES		Prohibition of the absolute input by the soft key (workpiece origin offset)
PROGRAM DELETE	○	Confirmation of the deletion of the program

Displayed item	Default	Corresponding function
ALL DATA DELETE	O	Confirmation of the deletion of all data
INPUT IN SETTING		Confirmation of a data update during the data setting process
UPDATE MODAL HIGHLIGHT DISPLAY	O	Display of updated modal information
AXIS STATUS DISPLAY	O	Axis status display
START FROM MIDDLE OF PROGRAM	O	Confirmation of the start from a middle block

6.6.3.2 Tool offset range setting screen

This screen displays the setting status of tool offset effective data ranges and lets you change their settings. (Hereinafter, the screen is referred to as the tool offset range setting screen.)
Up to 20 pairs of numbers can be specified to identify tool offset number ranges, and an effective offset value range can be defined for each of these 20 pairs.

Displaying and setting the tool offset range setting screen

Procedure

1 Press the  function key.

2 Press the continuous menu key  at the right edge of the screen several times until the [GUARD] soft key is displayed.

3 Press the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).

4 If any screen other than the tool offset range setting screen is displayed, press the [OFFSET] soft key. The tool offset range setting screen is displayed. What is displayed in this screen differs depending on the system configuration described later.

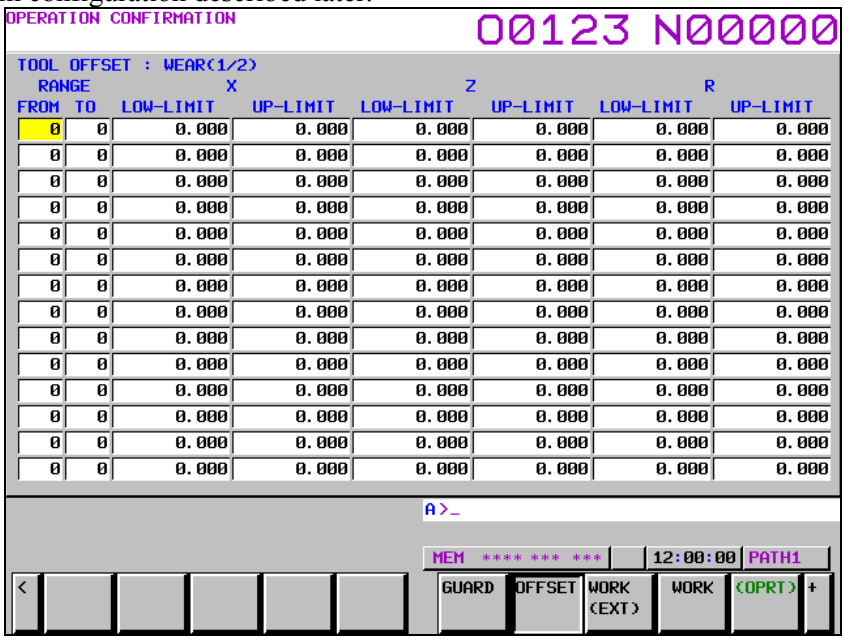


Fig. 6.6.3.2 (a) Tool offset range setting screen

5 Press the soft key [(OPRT)].

6 Move the cursor to the item you want to set, by using the  and  keys, , , , and  keys, or the [SWITCH] soft key.

7 Press the MDI key, enter necessary data, and then press the [INPUT] soft key.

If the set effective data range is invalid for any of the reasons listed below, the input data range check is not performed normally and the input data is rejected.

- There is a tool offset number overlap.
- The upper and lower limit values are reversed.
- The values are not effective (e.g., more pairs of offset numbers than allowed are set).
- Either of the tool offset numbers is 0.

Also, the input data range check is invalidated in the following cases.

- Both the upper and lower limit values for the tool offset number are 0.
- The upper and lower offset limit values are identical.

Explanation

- System configuration

What to set differs for each of the following system configurations:

M

- Tool offset memory A
- Tool offset memory B
- Tool offset memory C

T

- Without geometry and wear offset
- With geometry and wear offset

M

- What to set with tool offset memory A

With tool offset memory A, an effective data range is specified using the four items in the Table 6.6.3.2 (a).

Table 6.6.3.2 (a)

Displayed item		What to set
RANGE	FROM	Specify a tool offset number range.
	TO	
-	LOW-LIMIT	Specify a valid tool offset value range in connection with a specified tool offset number range.
	UP-LIMIT	

- What to set with tool offset memory B

With tool offset memory A, an effective data range is specified using the six items in the Table 6.6.3.2 (b).

Table 6.6.3.2 (b)

Displayed item		What to set
RANGE	FROM	Specify a tool offset number range.
	TO	
GEOM	LOW-LIMIT	Specify a valid tool offset value range for geometry in connection with a specified tool offset number range.
	UP-LIMIT	
WEAR	LOW-LIMIT	Specify a valid tool offset value range for wear in connection with a specified tool offset number range.
	UP-LIMIT	

- What to set with tool offset memory C

With tool offset memory C, an effective data range is specified using the ten items in the Table 6.6.3.2 (c).

Table 6.6.3.2 (c)

Displayed item			What to set
RANGE		FROM	Specify a tool offset number range.
		TO	
GEOM	LENGTH	LOW-LIMIT	Specify a valid tool offset value range for geometry length in connection with a specified tool offset number range.
		UP-LIMIT	
	RADIUS	LOW-LIMIT	Specify a valid tool offset value range for geometry radius in connection with a specified tool offset number range.
		UP-LIMIT	
WEAR	LENGTH	LOW-LIMIT	Specify a valid tool offset value range for wear length in connection with a specified tool offset number range.
		UP-LIMIT	
	RADIUS	LOW-LIMIT	Specify a valid tool offset value range for wear radius in connection with a specified tool offset number range.
		UP-LIMIT	

In the case of this configuration, all the information needed to set an input data range cannot be displayed in a single screen page. Set the information while switching pages using the [SWITCH] soft key. The screen provides an indication that lets you know which part of the information is currently displayed.

T

- What to set without geometry/wear offset

Without geometry/wear offset, an effective data range is specified using the eight items in the Table 6.6.3.2 (d).

Table 6.6.3.2 (d)

Displayed item		What to set
RANGE	FROM	Specify a tool offset number range.
	TO	
X	LOW-LIMIT	Specify a valid tool offset value range for the X-axis in connection with a specified tool offset number range.
	UP-LIMIT	
Z	LOW-LIMIT	Specify a valid tool offset value range for the Z-axis in connection with a specified tool offset number range.
	UP-LIMIT	
RADIUS	LOW-LIMIT	Specify a valid tool offset value range for tool-nose radius in connection with a specified tool offset number range.
	UP-LIMIT	

NOTE
The radius-related items are not displayed if the cutter or tool nose radius compensation is not used.

- What to set with geometry/wear offset

With geometry/wear offset, an effective data range is specified using the 12 items in the Table 6.6.3.2 (e).

Table 6.6.3.2 (e)

Displayed item			What to set
RANGE		FROM	Specify a tool offset number range.
		TO	
GEOM	X	LOW-LIMIT	Specify a valid tool offset value range for the geometry X-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	Z	LOW-LIMIT	Specify a valid tool offset value range for the geometry Z-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	RADIUS	LOW-LIMIT	Specify a valid tool offset value range for geometry tool-nose radius in connection with a specified tool offset number range.
		UP-LIMIT	
WEAR	X	LOW-LIMIT	Specify a valid tool offset value range for the wear X-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	Z	LOW-LIMIT	Specify a valid tool offset value range for the wear Z-axis in connection with a specified tool offset number range.
		UP-LIMIT	
	RADIUS	LOW-LIMIT	Specify a valid tool offset value range for wear tool-nose radius in connection with a specified tool offset number range.
		UP-LIMIT	

In the case of this system, all the information needed to set an input data range cannot be displayed in a single screen page. Set the information while switching pages using the [SWITCH] soft key. The screen provides an indication that lets you know which part of the information is currently displayed.

NOTE
The radius-related items are not displayed if the tool nose radius compensation option is not displayed.

- Example of setting an input data range

For example, suppose that the following values are set with offset memory A.

FROM : TO	LOW-LIMIT : UP-LIMIT
1 : 20	0.000 : 100.000

In this case, the tool offset input screen accepts only offset values from 0.000 to 100.000 for offset numbers 1 to 20.
If you attempt to input any other value, the warning message "DATA IS OUT OF RANGE" is displayed.

6.6.3.3 Workpiece origin offset range setting screen

This screen displays the setting status of workpiece origin offset and external workpiece origin offset effective data ranges and lets you change their settings. (Hereinafter, the screen is referred to as the workpiece origin offset range setting screen.)

Up to six pairs of values can be specified to identify workpiece coordinate ranges for the workpiece origin offset, and an effective offset value range can be defined for each of the axes of these six pairs.
As for the external workpiece origin offset, an effective offset value range can be specified for each axis.

Displaying and setting the workpiece origin offset range setting screen

Procedure

1 Press the  function key.
2 Press the continuous menu key  at the right edge of the screen several times until the [GUARD] soft key is displayed.
3 Press the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).
4 If any screen other than the workpiece origin offset range setting screen is displayed, press the [WORK] soft key. The workpiece origin offset range setting screen is displayed.

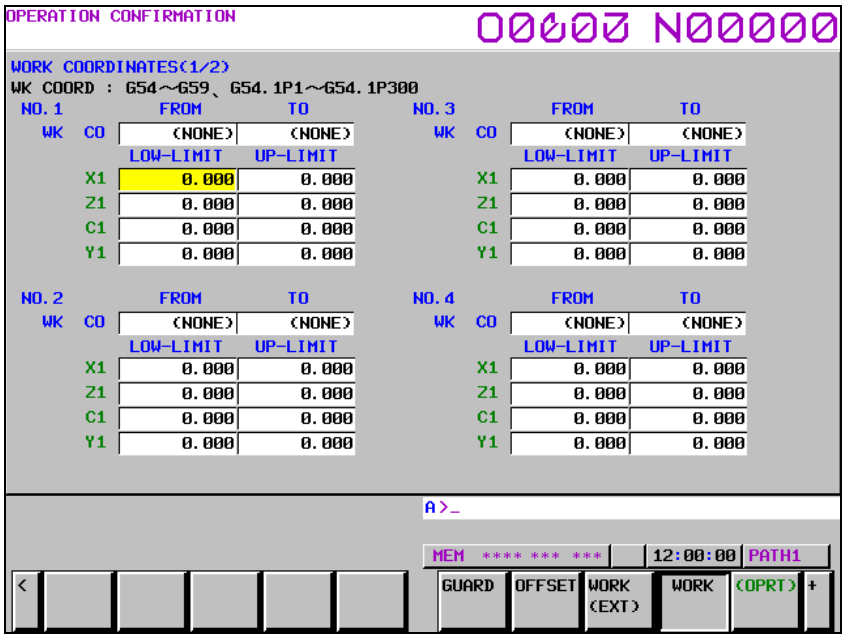


Fig. 6.6.3.3 (a) Workpiece origin offset range setting screen

5 Press the soft key [(OPRT)].

6 Move the cursor to the item you want to set, by using the PAGE ↑ and PAGE ↓ keys, or ↑, ↓, ←, and → keys.

7 Press the MDI key, enter necessary data, and then press the [INPUT] soft key.

If the set effective data range is invalid for any of the reasons listed below, the input data range check is not performed normally and the input data is rejected.

- There is a workpiece coordinate overlap.
- The upper and lower limit values are reversed.
- The values are not effective (e.g., an invalid workpiece coordinate system is set).
- The upper limit value is set for the workpiece coordinate system when 0 is set for the lower limit value.

Also, the input data range check is invalidated in the following cases.

- Both the upper and lower limit values for the workpiece coordinate system are 0.
- The upper and lower limit values for each offset are identical.

Explanation

- What to set for the workpiece origin offset

For the workpiece origin offset, an effective data range is specified using the four items in the Table 6.6.3.3 (a).

Table 6.6.3.3 (a)

Displayed item		What to set
RANGE	FROM	Specify a workpiece coordinate system range.
	TO	
AXIS NAME	LOW-LIMIT	Specify a valid offset value range in connection with a specified workpiece coordinate system range.
	UP-LIMIT	

- What to set for the external workpiece origin offset

For the external workpiece origin offset, an effective data range is specified using the two items in the Table 6.6.3.3 (b).

Table 6.6.3.3 (b)

Displayed item		What to set
AXIS NAME	LOW-LIMIT UP-LIMIT	Specify a valid external workpiece origin offset value range on each axis.

6.6.3.4 Y-axis tool offset range setting screen

T

In the case of a lathe system, this screen displays the setting status of Y-axis tool offset effective data ranges and lets you change their settings. (Hereinafter, the screen is referred to as the Y-axis tool offset range setting screen.)
Up to four pairs of values can be specified to identify Y-axis tool offset number ranges, and an effective offset value range can be defined for each of these four pairs.

Displaying and setting the Y-axis tool offset range setting screen

Procedure

1 Press the  function key.

2 Press the continuous menu key  at the right edge of the screen several times until the [GUARD] soft key is displayed.

3 Press the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).

4 If any screen other than the Y-axis tool offset range setting screen is displayed, press the [OFST.2] soft key. The Y-axis tool offset range setting screen is displayed. What is displayed in this screen differs depending on such factors as whether tool geometry/wear offsets are present.

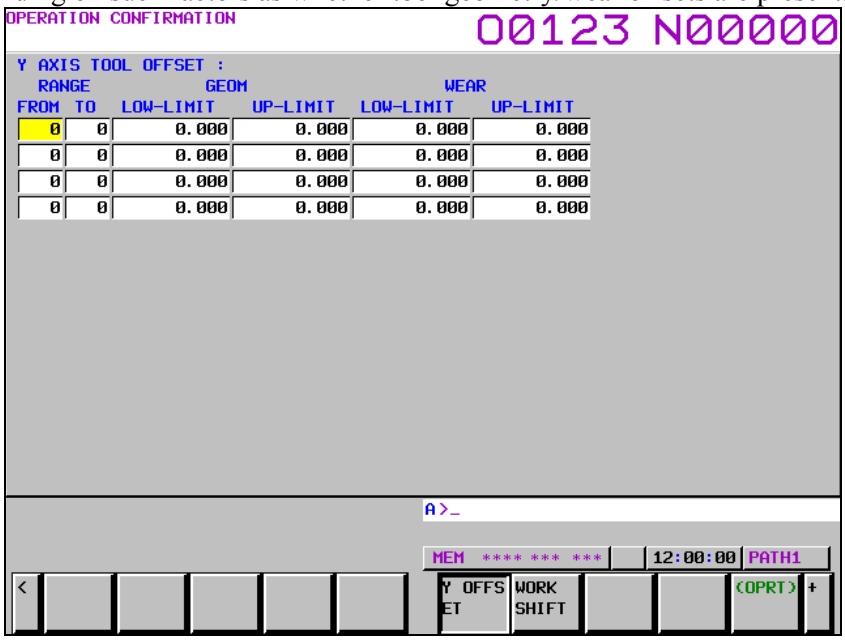


Fig. 6.6.3.4 (a) Y-axis tool offset range setting screen

5 Press the soft key [(OPRT)].

6 Move the cursor to the item you want to set, by using the  and  keys, , , , and  keys, or the [SWITCH] soft key.

7 Press the MDI key, enter necessary data, and then press the [INPUT] soft key.

If the set effective data range is invalid for any of the reasons listed below, the input data range check is not performed normally and the input data is rejected.

- There is a tool offset number overlap.
- The upper and lower limit values are reversed.
- The values are not effective (e.g., more pairs of offset numbers than allowed are set).
- Either of the tool offset numbers is 0.

Also, the input data range check is invalidated in the following cases.

- Both the upper and lower limit values for the tool offset number are 0.
- The upper and lower offset limit values are identical.

Explanation

- What to set without geometry/wear offsets

Without geometry/wear offsets, an effective data range is specified using the four items in the Table 6.6.3.4 (a).

Table 6.6.3.4 (a)

Displayed item		What to set
RANGE	FROM	Specify a Y-axis tool offset number range.
	TO	
-	LOW-LIMIT	Specify a valid tool offset value range in connection with a specified Y-axis tool offset number range.
	UP-LIMIT	

- What to set with geometry/wear offsets

With geometry/wear offsets, an effective data range is specified using the six items in the Table 6.6.3.4 (b).

Table 6.6.3.4 (b)

Displayed item		What to set
RANGE	FROM	Specify a Y-axis tool offset number range.
	TO	
GEOM	LOW-LIMIT	Specify a valid tool offset value range for geometry in connection with a specified Y-axis tool offset number range.
	UP-LIMIT	
WEAR	LOW-LIMIT	Specify a valid tool offset value range for wear in connection with a specified Y-axis tool offset number range.
	UP-LIMIT	

6.6.3.5 Workpiece shift range setting screen

T

In the case of a lathe system, this screen displays the setting status of shift effective data ranges of workpiece shift coordinate systems and lets you change their settings. (Hereinafter, the screen is referred to as the workpiece shift range setting screen.)
An offset value range can be specified for each axis.

Displaying and setting workpiece shift input ranges

Procedure

1 Press the function key .

2 Press the continuous menu key  at the right edge of the screen several times until the soft key [GUARD] is displayed.

3 Press the [GUARD] soft key. The setting screen that was displayed last with relation to any operation confirmation function is displayed (the operation confirmation function setting screen is the first such screen that appears after the system is restarted).

4 If any screen other than the workpiece shift range setting screen is displayed, press the [WORK SHIFT] soft key. The workpiece shift range setting screen is displayed.

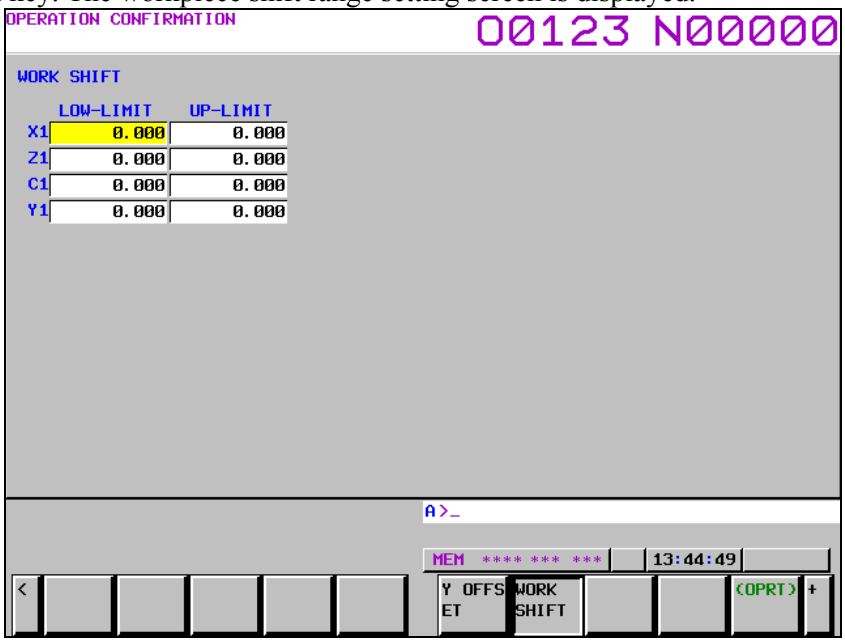


Fig. 6.6.3.5 (a) Workpiece shift range setting screen

5 Press the soft key [(OPRT)].

6 Move the cursor to the item you want to set, by using the PAGE ↑ and PAGE ↓ keys, or ↑, ↓, ←, and → keys.

7 Press the MDI key, enter necessary data, and then press the [INPUT] soft key.

If the set effective data range is invalid for any of the reasons listed below, the input data range check is not performed normally and the input data is rejected.

- The upper and lower limit values are reversed.

Also, the input data range check is invalidated in the following cases.

- The upper and lower offset limit values are identical.

Explanation

- What to set for the workpiece shift

For the workpiece shift, an effective data range is specified using the two items in the Table 6.6.3.5 (a).

Table 6.6.3.5 (a)

Displayed item		What to set
AXIS NAME	LOW-LIMIT	Specify a valid workpiece shift coordinate system shift value range on each axis.
	UP-LIMIT	

7 ALARM AND SELF-DIAGNOSIS FUNCTIONS

When an alarm occurs, the corresponding alarm screen appears to indicate the cause of the alarm. The causes of alarms are classified by error codes and number. Up to 60 previous alarms can be stored and displayed on the screen (alarm history display).
The system may sometimes seem to be at a halt, although no alarm is displayed. In this case, the system may be performing some processing. The state of the system can be checked using the self-diagnosis function.

7.1 ALARM DISPLAY

Explanation

- Alarm screen

When an alarm is issued, the display changes to the alarm screen. Two alarm screens "DETAIL" and "ALL PATH" are provided. You can choose one of the screens by pressing the corresponding soft key.

- Detail screen
  Alarm information for the currently selected path is displayed.

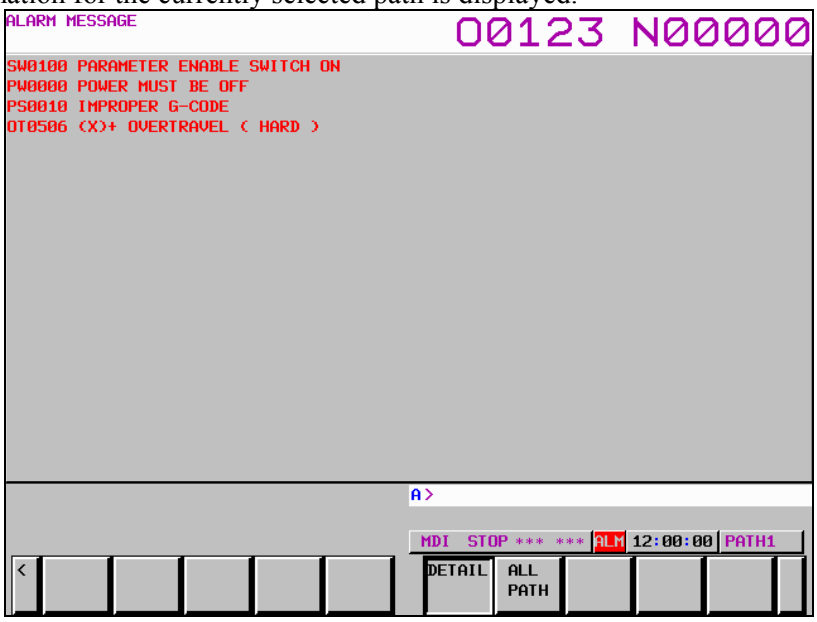


Fig. 7.1 (a) Alarm detail screen

- All path screen
  Alarm information for all paths is displayed sequentially from path 1.

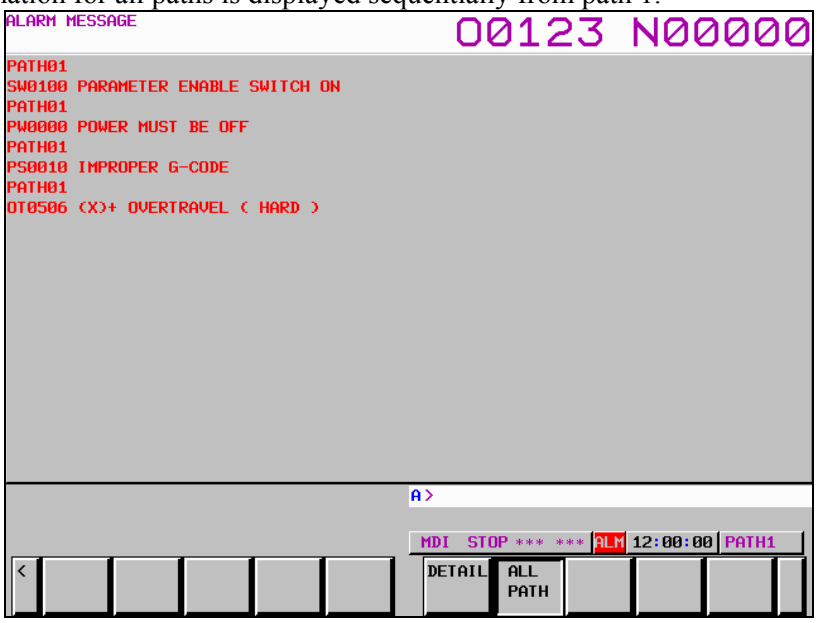


Fig. 7.1 (b) All path screen

- Displaying an alarm screen

ALM is sometimes indicated in the bottom part of the screen display without displaying an alarm screen.

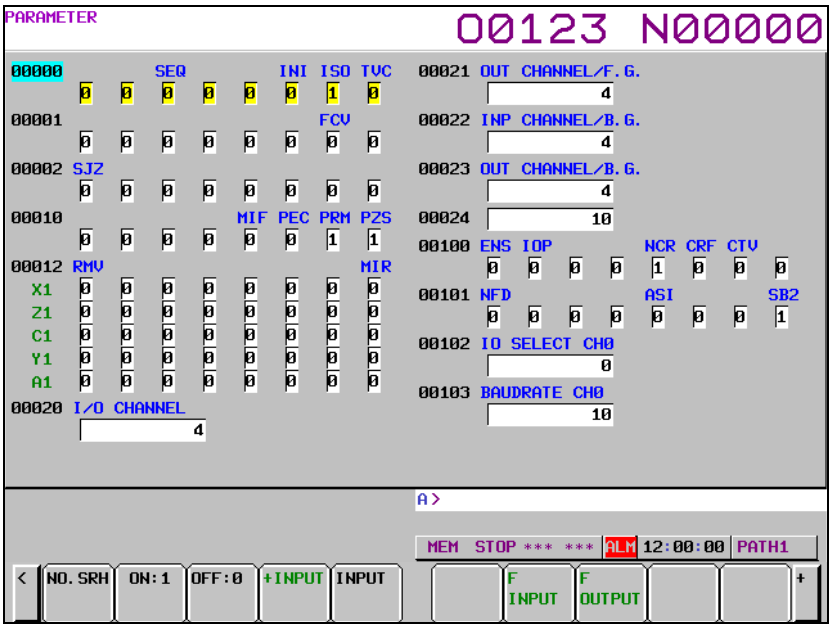


Fig. 7.1 (c) Parameter screen

In this case, display the alarm screen by following the steps below.

1 Press the function key 

.

2 Press the soft key [ALARM].

3 Pressing the [ALARM] soft key changes the screen display to the "DETAIL" screen (or the alarm screen selected previously), and the [DETAIL] and [ALL PATH] soft keys appears.
   - Pressing the [DETAIL] soft key displays the "DETAIL" screen.
   - Pressing the [ALL PATH] soft key displays the "ALL PATH" screen.

If the number of paths is 1, pressing the [ALARM] soft key displays the "DETAIL" screen, but the [ALARM] soft key indication remains unchanged.

4 You can change pages by using the page key.

- Releasing alarm

The cause of an alarm can be determined from the error code, number, and associated message. To release the alarm, generally correct the cause, then press the reset key.

- Error code and number

The type of an alarm is indicated by an error code and number.
Example: PS0010, SV0004, etc.
For details, see Appendix G, "ALARMS".

7.2 ALARM HISTORY DISPLAY

The CNC including the latest alarm are stored and displayed on the screen.
The display procedure is explained below.

NOTE

Alarms, operations, external operator messages, data change history data, etc. are stored in the same storage area. If the storage capacity is exceeded due to other history data, alarm history data may be erased. If, however, bit 2 (SAH) of parameter No. 11354 is 1, up to 50 alarms can be recorded regardless of the operation history state.
The maximum number of history data items that can be stored is as follows, where the history data is assumed to include alarm data only:
If bit 7 (HAL) of parameter No. 3196 is 1, about 4000 items.
If bit 7 (HAL) of parameter No. 3196 is 0, modal information and coordinates observed when an alarm is issued are also recorded, so that the maximum number varies from system to system.
(Example: For a 5-axis system, about 280 data items; for a 10-axis system, about 200 data items)
If bit 3 (EAH) of parameter No. 3112 is 1, however, macro and external alarm messages are also stored, so that the maximum number further decreases.

Alarm history display

Procedure

1 Press the function key 

.

2 Press the soft key [HISTRY].
An alarm history is displayed.
The following information is displayed:
<1> Date and time of alarm issuance
<2> Alarm type
<3> Alarm number
<4> Alarm message (sometimes not displayed depending on the alarm)
<5> Page No.

3 You can change pages by using the page key.

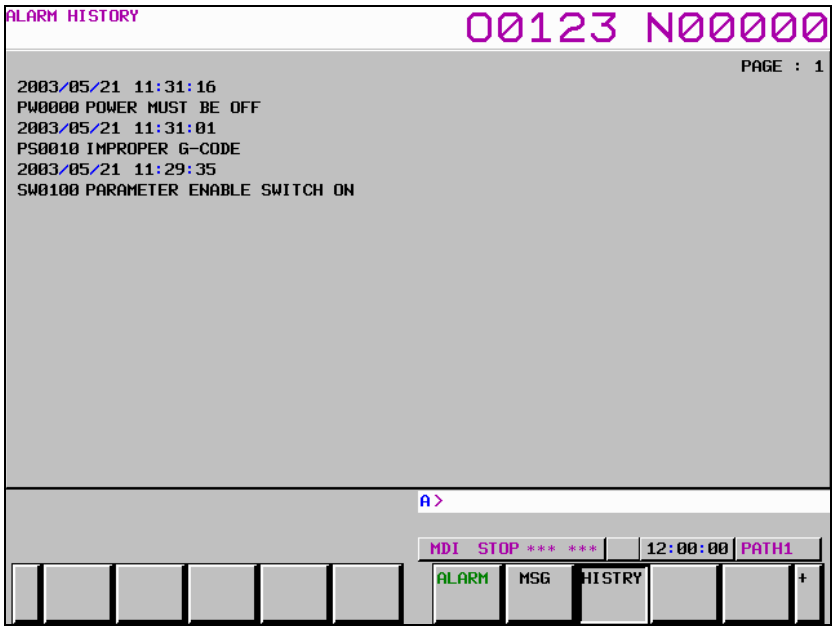


Fig. 7.2 (a) Alarm history screen

7.3 CHECKING BY DIAGNOSTIC DISPLAY

The system may sometimes seem to be at a halt, although no alarm has occurred. In this case, the system may be performing some processing. Diagnostic display can be used to check the system status.

Procedure for Diagnostic display

Procedure

1 Press the function key 

.

2 Press the soft key [DIAGNOSIS].

3 The diagnosis screen has more than 1 pages. Select the screen by the following operation.

(1) Change the page by the page change key.

(2) Method by soft key

- Key input the number of the diagnosis data to be displayed.
- Press the soft key [NO.SRH].

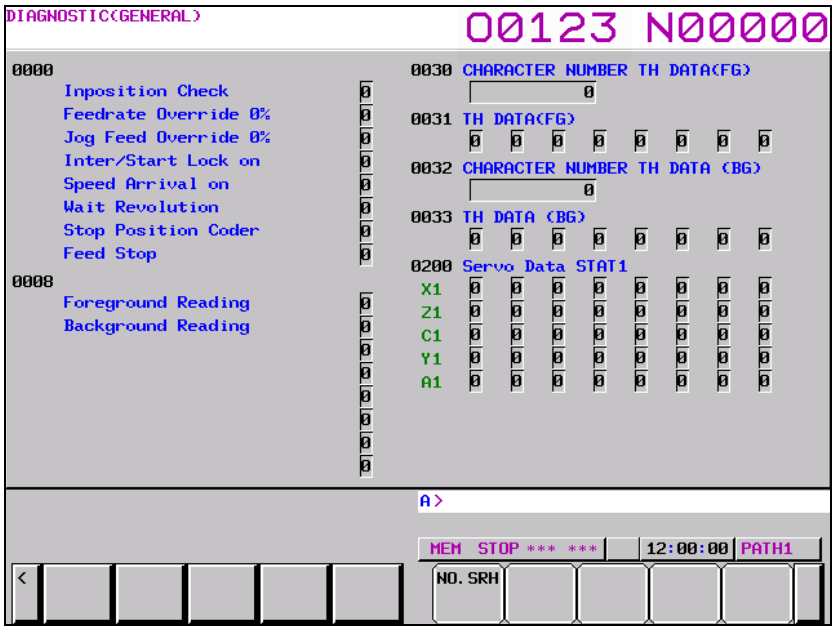


Fig. 7.3 (a) Diagnostic display

7.4 RETURN FROM THE ALARM SCREEN

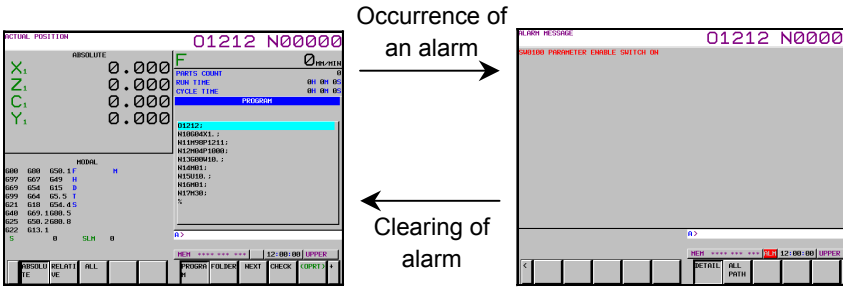
7.4.1 Return from the Alarm Screen

When alarms are cleared or function key MESSAGE is pressed on the alarm screen, the screen displayed before the alarm screen appears.
To enable this function, set bit 4 (ADC) of parameter No. 11302 is set to 1.

Switching between screens when alarms are cleared

When all alarms are cleared on the alarm screen, the screen displayed before the alarm screen appears again.
When the alarm screen was displayed automatically due to occurrence of an alarm, the screen displayed immediately before the alarm appears again.
When the alarm screen was displayed by pressing function key MESSAGE during occurrence of an alarm, the screen displayed immediately before the alarm appears again.

(Example)

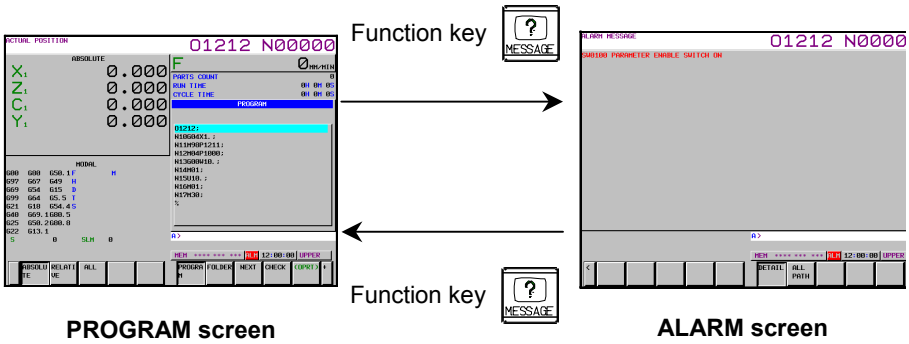


NOTE
Even if alarms are cleared when the alarm screen is not displayed, the current screen is not changed.

Switching between screens by the function key

When function key MESSAGE is pressed on the alarm screen, the screen displayed before the alarm screen appears. Press function key MESSAGE to switch to the alarm screen for checking for alarms and then press function key MESSAGE to return to the previous screen.

(Example)



If function key MESSAGE is pressed when the alarm screen was displayed automatically due to occurrence of an alarm, the screen displayed before the alarm screen appears again.

Restrictions

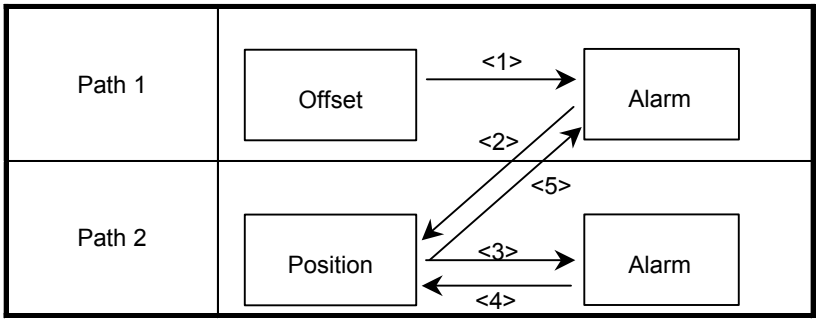
- Switching to the custom screen is not performed.
- The screens to which the alarm screen can be switched are only the screen selected by the chapter selection soft key.

7.4.2 Relationship with Other Functions

Relationship between the screen switching function and a return from the alarm screen during switching between paths

(1) When bit 5 (PSC) of parameter No. 3208 is set to 0, if paths are switched by the path switching signal, the screen last selected in the path appears again.
At this time, even if a return from the alarm screen to the previous screen is performed in one path, a return is not performed in the other path and the alarm screen remains displayed.

(Example)

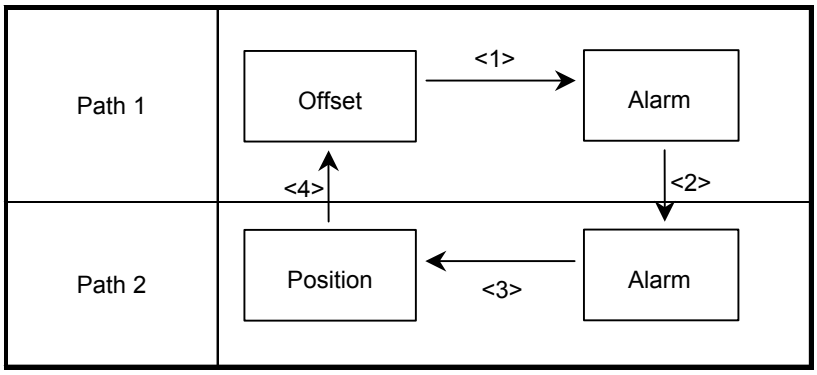


<1> When the message key is pressed on the offset screen of path 1, the alarm screen (path 1) appears.
<2> When switching to path 2 is performed on the alarm screen of path 1, the position screen of path 2 appears (when the screen last displayed in path 2 is the position screen).
<3> When the message key is pressed on the position screen of path 2, the alarm screen (path 2) appears.
<4> When the alarm is cleared or the message key is pressed on the alarm screen of path 2, a return to the position screen (path 2) is performed.
<5> When switching to path 1 is performed, the alarm screen (path 1) appears.

(2) When PSC of parameter No. 3208 is set to 1, if paths are switched by the path switching signal, the screen displayed immediately before the path switching appears again.

At this time, if a return from the alarm screen to the previous screen is performed in one path, the screen of the path in which a return was performed appears in the other path.

(Example)



<1> When the message key is pressed on the offset screen of path 1, the alarm screen (path 1) appears.
<2> When switching to path 2 is performed on the alarm screen of path 1, the alarm screen (path 2) appears.
<3> When the alarm is cleared on the alarm screen of path 2, the offset screen (path 2) appears.
<4> When switching to path 1 is performed, the offset screen of path 1 appears.

8 DATA INPUT/OUTPUT

Information stored in external I/O devices can be read into the CNC, and information can be written into external I/O devices.
External I/O devices include memory cards and USB memory that can be mounted to the memory card interface located on the left side of the display unit and personal computers and data servers that can be connected via embedded Ethernet.
The following types of data in the Table 8 (a) can be input and output.

Table 8 (a)

Data type	Default file name	Reference item
Program	ALL-PROG.TXT	8.2.1
Offset data	TOOLOFST.TXT	8.2.3
Parameter	CNC-PARA.TXT	8.2.2
Pitch error compensation data	PITCH.TXT	8.2.4
3-dimensional error compensation data	COMP3D.TXT	8.2.5
Three-dimensional rotary error compensation data	CMP3DROT.TXT	8.2.6
Custom macro common variable	MACRO.TXT	8.2.7
Workpiece coordinate system data	EXT_WKZ.TXT	8.2.8
Operation history data	OPRT_HIS.TXT	8.2.9
Tool management data (tool management function)	TOOL_MNG.TXT	8.2.10.1-2
Magazine data (tool management function)	MAGAZINE.TXT	8.2.10.3-4
Tool life status name (tool management function)	STATUS.TXT	8.2.10.5-6
Customize data name (tool management function)	CUSTOMIZ.TXT	8.2.10.7-8
Customize data displayed as tool management data (tool management expansion)	DISPCSTM.TXT	8.2.10.9-10
Spindle waiting position name (tool management expansion)	POSNAME.TXT	8.2.10.11-12
Decimal point position data of customize data (tool management expansion)	POINTPOS.TXT	8.2.10.13-14
Tool geometry data (tool management function for oversize tools)	TOOLGEOM.TXT	8.2.10.15-16
Workpiece setting error value	WSEC_VAL.TXT	8.2.11
Tool life management data	TOOLLIFE.TXT	8.2.12
Maintenance information	MAINTINF.TXT	
Periodic maintenance data (periodic maintenance: status screen)	MAINTENA.TXT	
Second geometry tool offset	SEC_GEOM.TXT	
Rotary table dynamic fixture offset	FIXTURE.TXT	
Real time custom macro variable	RTMMACRO.TXT	
System configuration data	SYS-CONF.TXT	
PMC signal protect	DIDOENBL.TXT	
Cause data (tool management tool attachment/detachment function)	TLCAUSE.TXT	
Attachment history data (tool management tool attachment/detachment function)	TLATAHIS.TXT	
Detachment history data (tool management tool attachment/detachment function)	TLDTAHIS.TXT	
Servo/spindle information	SV_SP_ID.TXT	
Machine system name data (periodic maintenance: machine system screen)	MAINTEMC.TXT	
Servo waveform diagnosis	WAVE-DGN.TXT	
Tool geometry data (check for tool post interference)	TOOL-FRM.TXT	
P code variable	PCODE.TXT	
Learning control data (learning control for parts cutting)	PRT_LN.TXT	
Learning control data (learning control)	LEARN.TXT	

The above types of data can be input and output on the screens for displaying and setting those types of data.
If NC data such as programs and parameters is to be written to a memory card or USB memory, and if a file with the same name already exists, it is possible to select whether to overwrite the existing file or cancel writing with appropriate operation.
This function is enabled by setting bit 1 (COW) of parameter No. 11308.

The external I/O device set in NC parameter No. 0020 is selected. See the Table 8 (b) below for details.

Table 8 (b)

Correspondence between settings and input/output units	
Setting	**Description**
0,1	RS-232-C serial port 1
2	RS-232-C serial port 2
4	Memory card interface
5	Data server interface
9	Embedded Ethernet interface
17	USB memory interface

NOTE
1 If the external input/output device is other than a data server interface or embedded Ethernet interface, a file name of up to 12 characters can be handled. NC programs on the memory of the CNC can each be assigned any file name of up to 32 characters, but the name is restricted to up to 12 characters in the input/output to and from external I/O devices.
2 If the external input/output device is a data server interface, embedded Ethernet interface, or USB memory interface, a file name of up to 32 characters can be handled.
3 When a memory card or USB memory interface is used as an external I/O device, up to 512 files can be handled.

8.1 OVERWRITING FILES ON A MEMORY CARD/USB MEMORY

Screen display

When an attempt is made to output NC data to a memory card, and if the specified file name or the default file name is the same as an existing file name on the memory card or USB memory, a confirmation message "OVERWRITE?" appears. (Bit 1 (COW) of parameter No. 11308 = 1)

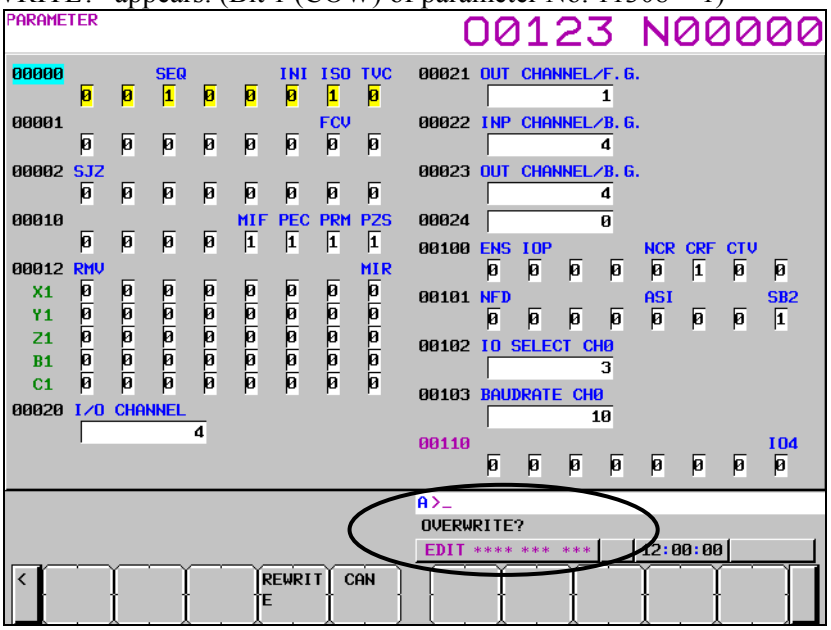


Fig. 8.1 (a) Screen display example

Procedure

On the output screen for the desired function, perform the following operation.

1 Press the soft key [F OUTPUT].
2 Press the soft key [EXEC]. If a file with the same name does not exist on the memory card or USB memory, the file is output in this step.
3 If a file with the same name exists on the memory card or USB memory, soft keys [REWRITE] and [CAN] appear.
Pressing the soft key [REWRITE] causes the file to be overwritten.
Pressing the soft key [CAN] causes output to be canceled.

Example) Output from the parameter screen (for 8.4/10.4-inch display unit)

1 Press the function key SYSTEM.
2 Press the soft key [PARAMETER].
3 Enter the EDIT mode or state emergency stop.
4 Press the soft key [(OPRT)].
5 For 8.4-inch display unit, press the continuous menu key ▷.
6 Press the soft key [F OUTPUT]. The soft key display switches from the one in Fig. 8.1 (b) to the one in Fig. 8.1 (c).
7 Press the soft key [EXEC]. Because no file name is specified, the file is output with a file name of CNC-PARA.TXT, but if a file with the same name exists on the memory card, the soft key display switches from the one in Fig. 8.1 (c) to the one in Fig. 8.1 (d), with a confirmation message appearing.
If a file with the same name does not exist on the memory card or USB memory, the file is output directly.

8 Pressing the soft key [REWRITE] causes the file to be overwritten.
Pressing the soft key [CAN] causes output to be canceled. If wishing to output the file after changing the file name, specify a file name after step 6, and perform step 7 again.



Fig. 8.1 (b) Soft key display before [F OUTPUT] is pressed

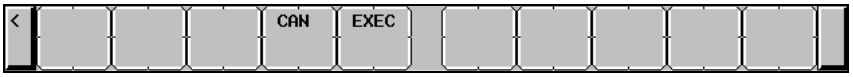


Fig. 8.1 (c) Soft key display after [F OUTPUT] is pressed



Fig. 8.1 (d) Soft key display after [EXEC] is pressed

Example) Output from the parameter screen (for 15/19-inch display unit)

1 Press the function key 

.

2 Press the vertical soft key [PARAMETER].
3 Enter the EDIT mode or state emergency stop.
4 Press the horizontal soft key [F OUTPUT]. The horizontal soft key display switches from the one shown in Fig. 8.1 (e) to the one in Fig. 8.1 (f).
5 Press the horizontal soft key [EXEC]. Because no file name is specified, the file is output with a file name of CNC-PARA.TXT, but if a file with the same name exists on the memory card or USB memory, the horizontal soft key display switches from the one in Fig. 8.1 (f) to the one in Fig. 8.1 (g), with a confirmation message appearing.
If a file with the same name does not exist on the memory card or USB memory, the file is output directly.
6 Pressing the horizontal soft key [REWRITE] causes the file to be overwritten.
Pressing the horizontal soft key [CAN] causes output to be canceled. If wishing to output the file after changing the file name, specify a file name after step 4, and perform step 5 again.



Fig. 8.1 (e) Soft key display before [F OUTPUT] is pressed



Fig. 8.1 (f) Soft key display after [F OUTPUT] is pressed



Fig. 8.1 (g) Soft key display after [EXEC] is pressed

⚠ CAUTION
For a memory card, even if soft key [REWRITE] is pressed, warning message “OVER WRITE FAILED” is displayed and the output is canceled in the following cases. For a USB memory, alarm SR1925 or SR1928 is displayed and the output is canceled.
- The file to be overwritten is for read only.
- The memory card is pulled out while the confirmation message appears.

8.2 INPUT/OUTPUT ON EACH SCREEN

This section explains how to input and output data of the following types to and from each operation screen: program, parameter, offset, pitch error compensation, 3-dimensional error compensation, three-dimensional rotary error compensation, macro variable, workpiece coordinate system data, operation history, and tool management.

Section 8.2, "INPUT/OUTPUT ON EACH SCREEN", consists of the following subsections:

8.2.1 Inputting and Outputting a Program....1174
8.2.1.1 Inputting a program....1174
8.2.1.2 Outputting a program....1176
8.2.1.3 Inputting and outputting of O8-digit....1177
8.2.2 Inputting and Outputting Parameters....1178
8.2.2.1 Inputting parameters....1178
8.2.2.2 Outputting parameters....1179
8.2.3 Inputting and Outputting Offset Data....1180
8.2.3.1 Inputting offset data....1180
8.2.3.2 Outputting offset data....1181
8.2.4 Inputting and Outputting Pitch Error Compensation Data....1185
8.2.4.1 Inputting pitch error compensation data....1185
8.2.4.2 Outputting pitch error compensation data....1186
8.2.4.3 Input/output format of pitch error compensation data....1187
8.2.5 Inputting and Outputting 3-dimensional Error Compensation Data....1188
8.2.5.1 Inputting 3-dimensional error compensation data....1188
8.2.5.2 Outputting 3-dimensional error compensation data....1190
8.2.5.3 Input/output format of 3-dimensional error compensation data....1190
8.2.6 Inputting and Outputting Three-dimensional Rotary Error Compensation Data....1192
8.2.6.1 Inputting three-dimensional rotary error compensation data....1192
8.2.6.2 Outputting three-dimensional rotary error compensation data....1193
8.2.6.3 Input/output format of three-dimensional rotary error compensation data....1194
8.2.7 Inputting and Outputting Custom Macro Common Variables....1196
8.2.7.1 Inputting custom macro common variables....1196
8.2.7.2 Outputting custom macro common variables....1197
8.2.8 Inputting and Outputting Workpiece Coordinates System Data....1199
8.2.8.1 Inputting workpiece coordinate system data....1199
8.2.8.2 Outputting workpiece coordinate system data....1200
8.2.9 Inputting and Outputting Operation History Data....1200
8.2.9.1 Outputting operation history data....1201
8.2.9.2 Inputting operation history signal selection data....1201
8.2.9.3 Outputting operation history signal section data....1202
8.2.9.4 Input/output format of operation history signal selection data....1203
8.2.10 Inputting and Outputting Tool Management Data....1204
8.2.10.1 Inputting tool management data....1204
8.2.10.2 Outputting tool management data....1205
8.2.10.3 Inputting magazine data....1206
8.2.10.4 Outputting magazine data....1208
8.2.10.5 Inputting tool life status name data....1208
8.2.10.6 Outputting tool life status name data....1209
8.2.10.7 Inputting name data of customize data....1210
8.2.10.8 Outputting name data of customize data....1211
8.2.10.9 Inputting customize data displayed as tool management data....1211
8.2.10.10 Outputting customize data displayed as tool management data....1212
8.2.10.11 Inputting spindle waiting position name data....1213

8.2.10.12 Outputting spindle waiting position name data 1214
8.2.10.13 Inputting decimal point position data of customize data 1215
8.2.10.14 Outputting decimal point position data of customize data 1216
8.2.10.15 Inputting tool geometry data 1217
8.2.10.16 Outputting tool geometry data 1218
8.2.11 Inputting and Outputting Workpiece Setting Error Compensation Value 1219
8.2.11.1 Inputting values on the workpiece setting error compensation screen 1219
8.2.11.2 Outputting values on the workpiece setting error compensation screen 1220
8.2.11.3 Input/output format of workpiece setting error values 1221
8.2.12 Inputting and Outputting Tool Life Management Data 1222
8.2.12.1 Inputting tool life management data 1222
8.2.12.2 Outputting tool life management data 1223

8.2.1 Inputting and Outputting a Program

8.2.1.1 Inputting a program

The following explains how to input a program from an external device to the memory of the CNC by using the program editing screen or program folder screen.

Inputting a program (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key  to display the program editing screen or program folder screen.
3 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
4 Press the soft key [(OPRT)].
5 Press the continuous menu key until soft key [F INPUT] appears.
Press the soft key [F INPUT].
6 Type the name of the file that you want to input.
Press the soft key [F SET]. To specify the program name to input, type the program name and press the soft key [P SET].
If the input file name is omitted, default input file name "ALL-PROG.TXT" is assumed.
For an explanation of the operations to be performed if an input file name [F SET] and an input program name [P SET] are omitted, see the Table 8.2.1.1 (a).
7 If you input the program, press the soft key [EXEC].
If you do not input the program, press the soft key [CAN]. Soft keys return to the operation selection soft keys.
When the input operation starts, "INPUT" blinks in the lower right part of the screen. When the operation ends, the "INPUT" indication disappears.
If you cancel the input during the operation, press the soft key [CANCEL].

Once read, the program is registered in the currently selected foreground folder.

NOTE
Foreground folders are displayed on the program folder screen. For an explanation of foreground folders, see II-12.1.3, and for how to change foreground folders, see III-11.6

Table 8.2.1.1 (a)

[F SET]	[P SET]	Input file name	Input program	Input program name
BLANK	BLANK	ALL-PROG.TXT	All programs in ALL-PROG.TXT	File name at the time the file is saved
BLANK	INPUT	ALL-PROG.TXT	First program in ALL-PROG.TXT	Program name set with [P SET]
INPUT	BLANK	File name set with [F SET]	All programs in the file specified with [F SET]	File name at the time the file is saved
INPUT	INPUT	File name set with [F SET]	First program in the file specified with [F SET]	Program name set with [P SET]

Inputting a program (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key 

.
3 Press the vertical soft key [PROGRAM] or [FOLDER] to display programs or a program directory.
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the continuous menu key  until horizontal soft key [F INPUT] appears.
Press the horizontal soft key [F INPUT].
6 Type the name of the file that you want to input.
Press the horizontal soft key [F SET]. To specify the program name to input, type the program name and press the horizontal soft key [P SET].
If the input file name is omitted, default input file name "ALL-PROG.TXT" is assumed.
For an explanation of the operations to be performed if an input file name [F SET] and an input program name [P SET] are omitted, see the Table 8.2.1.1 (b).
7 If you input the program, press the horizontal soft key [EXEC].
If you do not input the program, press the horizontal soft key [CAN]. Soft keys return to the operation selection soft keys.
When the input operation starts, "INPUT" blinks in the lower right part of the screen. When the operation ends, the "INPUT" indication disappears.
If you cancel the input during the operation, press the horizontal soft key [CANCEL].

Once read, the program is registered in the currently selected foreground folder.

NOTE
Foreground folders are displayed on the program folder screen. For an explanation of foreground folders, see II-12.1.3, and for how to change foreground folders, see III-11.6

Table 8.2.1.1 (b)

[F SET]	[P SET]	Input file name	Input program	Input program name
BLANK	BLANK	ALL-PROG.TXT	All programs in ALL-PROG.TXT	File name at the time the file is saved
BLANK	INPUT	ALL-PROG.TXT	First program in ALL-PROG.TXT	Program name set with [P SET]
INPUT	BLANK	File name set with [F SET]	All programs in the file specified with [F SET]	File name at the time the file is saved
INPUT	INPUT	File name set with [F SET]	First program in the file specified with [F SET]	Program name set with [P SET]

8.2.1.2 Outputting a program

A program stored in the memory of the CNC unit is output to an external device.

Outputting a program (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key  to display the program editing screen or program folder screen.
3 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
4 Press the soft key [(OPRT)].
5 Press the continuous menu key  until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
6 Type the program number to output and press the soft key [P SET]. To specify an output file name, type the output file name and press the soft key [F SET].
If, in this step, no output file name or no program name is specified, all programs in the foreground folders are output to "ALL-PROG.TXT".
For an explanation of the operations to be performed if an output file name [F SET] and an output program name [P SET] are omitted, see the Table 8.2.1.2 (a).
7 If you output the program, press the soft key [EXEC].
If you do not output the program, press the soft key [CAN]. Soft keys return to the operation selection soft keys.
When the output operation starts, "OUTPUT" blinks in the lower right part of the screen. When the operation ends, the "OUTPUT" indication disappears.
If you cancel the output during the operation, press the soft key [CANCEL].

NOTE
Foreground folders are displayed on the program folder screen. For an explanation of foreground folders, see II-12.1.3, and for how to change foreground folders, see III-11.6

Table 8.2.1.2 (a)

[F SET]	[P SET]	Output file name	Output program
BLANK	BLANK or (O-9999)	ALL-PROG.TXT	All programs in the foreground folders displayed in the program folder
BLANK	INPUT	Program name set with [P SET]	Program in the NC that is set with [P SET]
INPUT	BLANK or (O-9999)	File name set with [F SET]	All programs in the foreground folders displayed in the program folder
INPUT	INPUT	File name set with [F SET]	Program in the NC that is set with [P SET]

Outputting a program (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [PROGRAM] or [FOLDER] to display programs or a program directory.
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the continuous menu key  until horizontal soft key [F OUTPUT] appears.
Press the horizontal soft key [F OUTPUT].

6 Type the program number to output and press the horizontal soft key [P SET]. To specify an output file name, type the output file name and press the horizontal soft key [F SET].
If, in this step, no output file name or no program name is specified, all programs in the foreground folders are output to "ALL-PROG.TXT".
For an explanation of the operations to be performed if an output file name [F SET] and an output program name [P SET] are omitted, see the Table 8.2.1.2 (b).

7 If you output the program, press the horizontal soft key [EXEC].
If you do not output the program, press the horizontal soft key [CAN]. Soft keys return to the operation selection soft keys.
When the output operation starts, "OUTPUT" blinks in the lower right part of the screen. When the operation ends, the "OUTPUT" indication disappears.
If you cancel the output during the operation, press the horizontal soft key [CANCEL].

NOTE
Foreground folders are displayed on the program folder screen. For an explanation of foreground folders, see II-12.1.3, and for how to change foreground folders, see III-11.6

Table 8.2.1.2 (b)

[F SET]	[P SET]	Output file name	Output program
BLANK	BLANK or (O-9999)	ALL-PROG.TXT	All programs in the foreground folders displayed in the program folder
BLANK	INPUT	Program name set with [P SET]	Program in the NC that is set with [P SET]
INPUT	BLANK or (O-9999)	File name set with [F SET]	All programs in the foreground folders displayed in the program folder
INPUT	INPUT	File name set with [F SET]	Program in the NC that is set with [P SET]

8.2.1.3 Inputting and outputting of O8-digit

When the file name is not specified, the file name that is output and input is different according to the digit of the program number and the external input/output device.

- Outputting program
The program is output with the following file name.

Table 8.2.1.3 (c)

Program number	External input/output device	File name
7-digit or less (Example: 12345)	Excluding Fast Data Server or Fast Data Server excluding in the storage mode	"O" plus a 7-digit numeric (Example: O0012345)
	Fast Data Server in the storage mode	"O" plus a 8-digit numeric (Example: O00012345)
8-digit (Example: 12345678)	ALL	a 8-digit numeric (Example: 12345678)

- Inputting program
When the external input/output device is excluding Fast Data Server or Fast Data Server excluding in the storage mode, an input file name is "O"+7-digit numeric.
When Fast Data Server in the storage mode, an input file name is "O"+8-digit numeric.

8.2.2 Inputting and Outputting Parameters

8.2.2.1 Inputting parameters

Parameters are loaded into the memory of the CNC unit from an external device. The input format is the same as the output format. When a parameter is loaded which has the same data number as a parameter already registered in the memory, the loaded parameter replaces the existing parameter.

Inputting parameters (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the continuous menu key  until soft key [SETTING] appears.
Press the soft key [SETTING].
4 Press the MDI switch on the machine operator's panel or enter state emergency stop.
5 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data.
Alarm SW0100 appears.
6 Press the function key .
7 Press the soft key [PARAMETER], then the parameter screen appears.
8 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
9 Press the soft key [(OPRT)].
10 Press the continuous menu key  until soft key [F INPUT] appears.
Press the soft key [F INPUT].
11 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "CNC-PARA.TXT" is assumed.
12 Press the soft key [EXEC].
This starts reading the parameter, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the parameter, press the soft key [CAN].
13 Press the function key .
14 Press the soft key [SETTING].
15 Press the MDI switch on the machine operator's panel or enter state emergency stop.
16 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
17 Turn the power of the CNC on again.

Inputting parameters (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [SETTING] appears.
Press the vertical soft key [SETTING].
4 Press the MDI switch on the machine operator's panel or enter state emergency stop.
5 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data.
Alarm SW0100 appears.
6 Press the function key .
7 Press the vertical soft key [NEXT PAGE] until vertical soft key [PARAMETER] appears.
Press the vertical soft key [PARAMETER], then the parameter screen appears.

8 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
9 Press the horizontal soft key [F INPUT].
10 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "CNC-PARA.TXT" is assumed.
11 Press the horizontal soft key [EXEC].
This starts reading the parameter, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the parameter, press the horizontal soft key [CAN].
12 Press the function key .
13 Press the vertical soft key [SETTING].
14 Press the MDI switch on the machine operator's panel or enter state emergency stop.
15 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
16 Turn the power of the CNC on again.

8.2.2.2 Outputting parameters

All parameters are output in a defined output format from the memory of the CNC to an external device.

Outputting parameters (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the soft key [PARAMETER], then the parameter screen appears.
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key  until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name "CNC-PARA.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the parameter, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting parameters (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [PARAMETER] appears.
Press the vertical soft key [PARAMETER], then the parameter screen appears.
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the horizontal soft key [F OUTPUT].
6 Type the file name that you want to output.
If the file name is omitted, default file name "CNC-PARA.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the parameter, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Explanation

- **Suppressing output of parameters set to 0**

When bit 1 (PRM) of parameter No. 0010 is set to 1, and soft key [EXEC] is pressed, the parameters in the Table 8.2.2.2 (a) are not output:

Table 8.2.2.2 (a)

	Other than axis type	Axis type
Bit type	Parameter for which all bits are set to 0.	Parameter for an axis for which all bits are set to 0.
Value type	Parameter whose value is 0.	Parameter for an axis for which the value is 0.

8.2.3 Inputting and Outputting Offset Data

8.2.3.1 Inputting offset data

Offset data is loaded into the memory of the CNC from an external device. The input format is the same as for offset value output. When an offset value is loaded which has the same offset number as an offset number already registered in the memory, the loaded offset data replaces existing data.

Inputting offset data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the continuous menu key until soft key [OFFSET] appears.
Press the soft key [OFFSET].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key  until soft key [F INPUT] appears.
Press the soft key [F INPUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "TOOLOFST.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the offset data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the offset data, press the soft key [CAN].

Inputting offset data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [OFFSET] appears.
Press the vertical soft key [OFFSET].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the horizontal soft key [F INPUT].
6 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "TOOLOFST.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts reading the offset data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the offset data, press the horizontal soft key [CAN].

8.2.3.2 Outputting offset data

All offset data is output in a defined output format from the memory of the CNC to an external device.

Outputting offset data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the continuous menu key  until soft key [OFFSET] appears.
Press the soft key [OFFSET].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key  until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name "TOOLOFST.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the offset data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting offset data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [OFFSET] appears.
Press the vertical soft key [OFFSET].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the horizontal soft key [F OUTPUT].
6 Type the file name that you want to output.
If the file name is omitted, default file name "TOOLOFST.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the offset data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Explanation

- Output format

Output format is as follows:

M

- Tool compensation memory A

```
%
G10 G90 P01 R_ Q_
G10 G90 P02 R_ Q_
 ...
G10 G90 P_ R_
%
```

Q_ : Virtual tool nose number (TIP). Not output when the virtual tool nose direction is not used.
P_ : Tool offset number (1 to the number of tool compensation pairs)
R_ : Tool compensation data. Output with a decimal point in the input unit used at output.

- Tool compensation memory B

```
%
G10 G90 L10 P01 R_ Q_
G10 G90 L11 P01 R_
G10 G90 L10 P02 R_ Q_
 ...
G10 G90 L11 P_ R_
%
```

L10 : Geometric compensation amount
L11 : Wear compensation amount
Q_, P_, and R_ have the same meanings as for tool compensation memory A.

- Tool compensation memory C

```
%
G10 G90 L10 P01 R_ Q_
G10 G90 L11 P01 R_
G10 G90 L12 P01 R_
G10 G90 L13 P01 R_
G10 G90 L10 P02 R_ Q_
 ...
G10 G90 L12 P_ R_
G10 G90 L13 P_ R_
%
```

L10 : Geometry compensation amount corresponding to the H code
L11 : Wear compensation amount corresponding to the H code
L12 : Geometry compensation amount corresponding to the D code
L13 : Wear compensation amount corresponding to the D code
Q_, P_, and R_ have the same meanings as for tool compensation memory A.

- If the cutting point command option is enabled

```
%
G10 G90 L10 P01 R_ Q_
G10 G90 L11 P01 R_
G10 G90 L12 P01 R_
G10 G90 L13 P01 R_
G10 G90 L110 P01 R_
G10 G90 L111 P01 R_
G10 G90 L10 P02 R_ Q_
 ...
G10 G90 L110 P01 R_
G10 G90 L111 P01 R_
%
```

L10 : Geometry compensation amount corresponding to the H code
L11 : Wear compensation amount corresponding to the H code
L12 : Geometry compensation amount corresponding to the D code (tool compensation)
L13 : Wear compensation amount corresponding to the D code (tool compensation)
L110 : Geometry compensation amount corresponding to the D code (corner R offset)
L111 : Wear compensation amount corresponding to the D code (corner R offset)
Q_, P_, and R_ have the same meanings as for tool compensation memory A.

T

The tool compensation amount and tool nose radius compensation amount are output in the following format.

```
%
G10 P01 X_ Z_ R_ Q_ Y_
G10 P02 X_ Z_ R_ Q_ Y_
...
G10 P__ X_ Z_ R_ Q_ Y_
G10 P10001 X_ Z_ R_ Y_
G10 P10002 X_ Z_ R_ Y_
...
G10 P100__ X_ Z_ R_ Y_
%
```

P_: Tool compensation number (1 to the number of tool compensation pairs)
Tool offset number: Specification of the tool compensation amount and tool wear compensation amount
10000 + tool offset number: Specification of the tool geometry compensation amount
X_: Tool compensation data (X). Output with a decimal point in the input unit used at output.
Z_: Tool compensation data (Z). Same as X_.
R_: Tool nose radius offset amount (R). The data format is the same as for X_.
When tool nose radius compensation is not provided, this item is not output.
Q_: Virtual tool nose number (TIP). When tool nose radius compensation is not provided, this item is not output.
Y_ : Tool compensation data (Y). The data format is the same as for X_.
When no Y-axis offset is provided, this item is not output.

The second geometry tool offset data is output in the following format.

```
%
G10 P20001 X_ Z_ Y_
G10 P20002 X_ Z_ Y_

G10 P200__ X_ Z_ Y_
%
```

P_ : Tool compensation number (1 to the number of tool compensation pairs)
Tool offset number: Specification of the tool compensation amount and tool wear compensation amount
20000 + tool offset number: Specification of the second tool geometry compensation amount
The other addresses are the same as for the tool compensation amount.

NOTE
The input format and output format do not depend on the G-code system A/B/C.

8.2.4 Inputting and Outputting Pitch Error Compensation Data

8.2.4.1 Inputting pitch error compensation data

Pitch error compensation data are loaded into the memory of the CNC from an external device. The input format is the same as the output format. When a pitch error compensation data is loaded which has the corresponding data number as a pitch error compensation data already registered in the memory, the loaded data replaces the existing data.

Inputting pitch error compensation data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the continuous menu key  until soft key [SETTING] appears.
Press the soft key [SETTING].
4 Press the MDI switch on the machine operator's panel or enter state emergency stop.
5 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data.
Alarm SW0100 appears.
6 Press the function key .
7 Press the continuous menu key  until soft key [PITCH ERROR] appears.
Press the soft key [PITCH ERROR].
8 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
9 Press the soft key [(OPRT)].
10 Press the continuous menu key  until soft key [F INPUT] appears.
Press the soft key [F INPUT].
11 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "PITCH.TXT" is assumed.
12 Press the soft key [EXEC].
This starts reading the pitch error compensation data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the pitch error compensation data, press the soft key [CAN].
13 Press the function key .
14 Press the soft key [SETTING].
15 Press the MDI switch on the machine operator's panel or enter state emergency stop.
16 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
17 Turn the power of the CNC on again.

Inputting pitch error compensation data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [SETTING] appears.
Press the vertical soft key [SETTING].
4 Press the MDI switch on the machine operator's panel or enter state emergency stop.
5 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data.
Alarm SW0100 appears.
6 Press the function key .

7 Press the vertical soft key [NEXT PAGE] until vertical soft key [PITCH ERROR] appears.
Press the vertical soft key [PITCH ERROR].
8 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
9 Press the horizontal soft key [F INPUT].
10 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "PITCH.TXT" is assumed.
11 Press the horizontal soft key [EXEC].
This starts reading the pitch error compensation data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the pitch error compensation data, press the horizontal soft key [CAN].
12 Press the function key .
13 Press the vertical soft key [SETTING].
14 Press the MDI switch on the machine operator's panel or enter state emergency stop.
15 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
16 Turn the power of the CNC on again.

8.2.4.2 Outputting pitch error compensation data

All pitch error compensation data are output in a defined output format from the memory of the CNC to an external device.

Outputting pitch error compensation data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the continuous menu key  until soft key [PITCH ERROR] appears.
Press the soft key [PITCH ERROR].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key  until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name "PITCH.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the pitch error compensation data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting pitch error compensation data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [SETTING] appears.
Press the vertical soft key [PITCH ERROR].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the horizontal soft key [F OUTPUT].
6 Type the file name that you want to output.
If the file name is omitted, default file name "PITCH.TXT" is assumed.

7 Press the horizontal soft key [EXEC].
This starts outputting the pitch error compensation data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.4.3 Input/output format of pitch error compensation data

Pitch error compensation data is input and output in the following input and output formats.

- Keywords

The following alphabets are used as keywords.
The numeric value following each keyword has the meaning listed below:

Table 8.2.4.3 (a)

Keyword	Meaning of the following numeric value
N	Pitch error compensation data number + 10000
Q	Data identification (1 : Parameter data, 0 : Pitch error compensation data)
L	Input / Output format of pitch error compensation data (Without L : Incremental value type, L1 : Total value type)
P	Pitch error compensation data value

- Format

The input / output format of pitch error compensation data is changed depending on bit 0 (APE) of parameter No. 3602.

In case of incremental value type (bit 0 (APE) of parameter No. 3602 is set to 0).

N	*****	Q0	P	****	;

In case of total value type (bit 0 (APE) of parameter No. 3602 is set to 1).

N	*****	Q0	L1	P	****	;

The 5-digit numeric value following N indicates a pitch error compensation data number to which a value of 10000 is added.
If the Bi-directional pitch error compensation is valid, the 5-digit numeric value following N indicates a pitch error compensation data number to which a value of 20000 is added.
Q0 indicates pitch error compensation data
L1 indicates that the input / output format of pitch error compensation data is total value type. When the input / output format is incremental type (bit 0 (APE) of parameter No. 3602 is set to 0), L1 is not to be specified. If the data with L1 is input when bit 0 (APE) of parameter No. 3602 is set to 0, or the data without L1 is input when bit 0 (APE) of parameter No. 3602 is set to 1, alarm SR1300 “ILLEGAL ADDRESS” is caused.
The numeric value following P indicates the value (integer value) of pitch error compensation data between -128 and 127.
The semicolon (;) indicates the end of block (LF in the ISO code or CR in the EIA code).

Example1 (bit 0 (APE) of parameter No.3602 is set to 0)
N10001Q0P100;
Pitch error compensation data number 1
Pitch error compensation data value 100
Example2 (bit 0 (APE) of parameter No.3602 is set to 1)
N10001Q0L1P100;
Pitch error compensation data number 1
Pitch error compensation data value 100

- Beginning and end of a record

A pitch error compensation data record begins with % and ends with %.

Example
% ;.....Beginning of record
N10000Q0P10;
N10001Q0P100;
:
N11279Q0P0;
%.....End of record

When parameters and pitch error compensation data are integrated into one file, % is added to the beginning and end of the file.

⚠ WARNING
1 Do not fail to perform reference position return manually after changing the pitch error compensation data or backlash compensation data. Without this, the machine position can deviate from the correct position.
2 If the compensation data is changed when the input format is total value type(bit 0 (APE) of parameter No. 3602 is set to 1), power must be turned off before operation is continued.

8.2.5 Inputting and Outputting 3-dimensional Error Compensation Data

8.2.5.1 Inputting 3-dimensional error compensation data

3-dimensional error compensation data are loaded into the memory of the CNC from an external device. The input format is the same as the output format. When a 3-dimensional error compensation data is loaded which has the corresponding data number as a 3-dimensional error compensation data already registered in the memory, the loaded data replaces the existing data.

Inputting 3-dimensional error compensation data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the continuous menu key until soft key [SETTING] appears.
Press the soft key [SETTING].
4 Press the MDI switch on the machine operator’s panel or enter state emergency stop.

5 Enter 1 in response to the prompt for “PARAMETER WRITE” in setting data.
Alarm SW0100 appears.
6 Press the function key .
7 Press the continuous menu key  until soft key [3D ERR COMP] appears.
Press the soft key [3D ERR COMP].
8 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
9 Press the soft key [(OPRT)].
10 Press the continuous menu key  until soft key [F INPUT] appears.
Press the soft key [F INPUT].
11 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “COMP3D.TXT” is assumed.
12 Press the soft key [EXEC].
This starts reading the 3-dimensional error compensation data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the 3-dimensional error compensation data, press the soft key [CAN].
13 Press the function key .
14 Press the soft key [SETTING].
15 Press the MDI switch on the machine operator’s panel or enter state emergency stop.
16 Enter 0 in response to the prompt for “PARAMETER WRITE” in setting data.
17 Turn the power of the CNC on again.

Inputting 3-dimensional error compensation data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [SETTING] appears.
Press the vertical soft key [SETTING].
4 Press the MDI switch on the machine operator’s panel or enter state emergency stop.
5 Enter 1 in response to the prompt for “PARAMETER WRITE” in setting data.
Alarm SW0100 appears.
6 Press the function key .
7 Press the vertical soft key [NEXT PAGE] until vertical soft key [3D ERR COMP] appears.
Press the vertical soft key [3D ERR COMP].
8 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
9 Press the horizontal soft key [F INPUT].
10 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “COMP3D.TXT” is assumed.
11 Press the horizontal soft key [EXEC].
This starts reading the 3-dimensional error compensation data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the 3-dimensional error compensation data, press the horizontal soft key [CAN].
12 Press the function key .
13 Press the vertical soft key [SETTING].
14 Press the MDI switch on the machine operator’s panel or enter state emergency stop.
15 Enter 0 in response to the prompt for “PARAMETER WRITE” in setting data.
16 Turn the power of the CNC on again.

8.2.5.2 Outputting 3-dimensional error compensation data

All 3-dimensional error compensation data are output in a defined output format from the memory of the CNC to an external device.

Outputting 3-dimensional error compensation data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key SYSTEM.
3 Press the continuous menu key ▷ until soft key [3D ERR COMP] appears.
Press the soft key [3D ERR COMP].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key ▷ until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name "COMP3D.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the 3-dimensional error compensation data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting 3-dimensional error compensation data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key SYSTEM.
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [3D ERR COMP] appears.
Press the vertical soft key [3D ERR COMP].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the horizontal soft key [F OUTPUT].
6 Type the file name that you want to output.
If the file name is omitted, default file name "COMP3D.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the 3-dimensional error compensation data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.5.3 Input/output format of 3-dimensional error compensation data

3-dimensional error compensation data is input and output in the following input and output formats.

- Keywords

The following alphabets are used as keywords.
The numeric value following each keyword has the meaning listed below:

Table 8.2.5.3 (a)

Keyword	Meaning of the following numeric value
N	Data number (compensation point number + 100000) as represented with six digits
A1	1st compensation axis

Keyword	Meaning of the following numeric value
A2	2nd compensation axis
A3	3rd compensation axis
P	Compensation data (-128 to 127)

- Format

3-dimensional error compensation data is output in the following format:

N	*****	A1	P	****	A2	P	****	A3	P	****	;

The 6-digit numeric value following N indicates a 3-dimensional error compensation data number to which a value of 100000 is added.
The numeric value following P indicates the value (integer value) of 3-dimensional error compensation data between -128 and 127.
The semicolon (;) indicates the end of block (LF in the ISO code or CR in the EIA code).

Example

```
N100001A1P100A2P110A3P120;
   3-dimensional error compensation data number:                  1
   Error compensation data value for the 1st compensation axis:  100
   Error compensation data value for the 2nd compensation axis: 110
   Error compensation data value for the 3rd compensation axis:  120
```

- Beginning and end of a record

A 3-dimensional error compensation data record begins with % and ends with %.

Example

```
% ;.....Beginning of record
N100001 A1 P1 A2 P2 A3 P3 ;
N100002 A1 P0 A2 P0 A3 P-3 ;
    :
N115625 A1 P1 A2 P1 A3 P0 ;
%.....End of record
```

- Input of compensation data using G10

Compensation data can be changed from a machining program, using the programmable parameter input function.

The command format is as follows:

```
%
G10 L51 ;
N_ P_ R_ ;
N_ P_ R_ ;
:
G11 ;
%
```

G10 L51: 3-dimensional error compensation data input mode
G11 : Cancellation of 3-dimensional error compensation data input mode
N : Compensation point number (1-15625)
P : Compensation axis number (1-3)
R : Compensation data (-128-127)

NOTE
1 In 3-dimensional error compensation data input mode, no other NC statements can be issued.
2 The decimal point cannot be used in address N, P, and R.

8.2.6 Inputting and Outputting Three-dimensional Rotary Error Compensation Data

8.2.6.1 Inputting three-dimensional rotary error compensation data

Three-dimensional rotary error compensation data are loaded into the memory of the CNC from an external device. The input format is the same as the output format. When a three-dimensional rotary error compensation data is loaded to a number where an old three-dimensional rotary error compensation data already exists, the newly loaded data replaces the existing data.

Inputting three-dimensional rotary error compensation data (for 8.4/10.4-inch display unit)

Procedure

1 Make the input device ready for reading.
2 Press the function key .
3 Press the continuous menu key  until soft key [SETTING] appears.
Press the soft key [SETTING].
4 Place the CNC in the MDI mode or the emergency stop state.
5 Enter 1 to “PARAMETER WRITE” in setting data.
Alarm SW0100 appears.
6 Press the function key .
7 Press the continuous menu key  until soft key [3D ERR ROT] appears.
Press the soft key [3D ERR ROT].
8 Place the CNC in the EDIT mode or the emergency stop state.
9 Press the soft key [(OPRT)].
10 Press the continuous menu key  until soft key [F INPUT] appears.
Press the soft key [F INPUT].
11 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “CMP3DROT.TXT” is assumed.

12 Press the soft key [EXEC].
This starts reading the three-dimensional rotary error compensation data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the three-dimensional rotary error compensation data, press the soft key [CAN].
13 Press the function key .
14 Press the soft key [SETTING].
15 Place the CNC in the MDI mode or the emergency stop state.
16 Enter 0 to "PARAMETER WRITE" in setting data.
17 Turn off and on the power of the CNC again.

Inputting three-dimensional rotary error compensation data (for 15/19-inch display unit)

Procedure

1 Make the input device ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [SETTING] appears.
Press the vertical soft key [SETTING].
4 Place the CNC in the MDI mode or the emergency stop state.
5 Enter 1 to "PARAMETER WRITE" in setting data.
Alarm SW0100 appears.
6 Press the function key .
7 Press the vertical soft key [NEXT PAGE] until vertical soft key [3D ERR ROT] appears.
Press the vertical soft key [3D ERR ROT].
8 Place the CNC in the EDIT mode or the emergency stop state.
9 Press the horizontal soft key [F INPUT].
10 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "CMP3DROT.TXT" is assumed.
11 Press the horizontal soft key [EXEC].
This starts reading the three-dimensional rotary error compensation data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the three-dimensional rotary error compensation data, press the horizontal soft key [CAN].
12 Press the function key .
13 Press the vertical soft key [SETTING].
14 Place the CNC in the MDI mode or the emergency stop state.
15 Enter 0 to "PARAMETER WRITE" in setting data.
16 Turn off and on the power of the CNC again.

8.2.6.2 Outputting three-dimensional rotary error compensation data

All three-dimensional rotary error compensation data are output in the output format from the memory of the CNC to an external device.

Outputting three-dimensional rotary error compensation data (for 8.4/10.4-inch display unit)

Procedure

1 Make the output device ready for output.

2 Press the function key SYSTEM.
3 Press the continuous menu key ▷ until soft key [3D ERR ROT] appears.
Press the soft key [3D ERR ROT].
4 Place the CNC in the EDIT mode or the emergency stop state.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key ▷ until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name “CMP3DROT.TXT” is assumed.
8 Press the soft key [EXEC].
This starts outputting the three-dimensional rotary error compensation data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting three-dimensional rotary error compensation data (for 15/19-inch display unit)

Procedure

1 Make the output device ready for output.
2 Press the function key SYSTEM.
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [3D ERR ROT] appears.
Press the vertical soft key [3D ERR ROT].
4 Place the CNC in the EDIT mode or the emergency stop state.
5 Press the horizontal soft key [F OUTPUT].
6 Type the file name that you want to output.
If the file name is omitted, default file name “CMP3DROT.TXT” is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the three-dimensional rotary error compensation data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.6.3 Input/output format of three-dimensional rotary error compensation data

Three-dimensional rotary error compensation data is input and output in the following input and output formats.

- Keywords

The following alphabets are used as keywords.
The numeric value following each keyword has the meaning listed below:

Table8.2.6.3 (a)

Keyword	Meaning of the following numeric value
N	Data number (compensation point number + 200000) as represented with six digits
A1	Translational error compensation value (C*nx*)
A2	Translational error compensation value (C*ny*)
A3	Translational error compensation value (C*nz*)
A4	Rotary error compensation value (C$n\alpha$)
A5	Rotary error compensation value (C$n\beta$)
A6	Rotary error compensation value (C$n\gamma$)

Keyword	Meaning of the following numeric value
P	Compensation data (-128 to 127)

- Format

Three-dimensional rotary error compensation data is output in the following format:

N	*****	A1	P	****	A2	P	****	A3	P	****	A4	P	****	A5	P	****	A6	P	****	;

The 6-digit numeric value following N indicates the number of <a three-dimensional rotary error compensation data number + 200000>.
The numeric value following P indicates the value (integer value) of three-dimensional rotary error compensation data between -128 and 127.
The semicolon (;) indicates the end of block (LF in the ISO code or CR in the EIA code).

Example

N200001A1P1A2P2A3P3A4P4A5P5A6P-1;

Three-dimensional rotary error compensation data number:	1
Translational error compensation data value (C*nx*):	1
Translational error compensation data value (C*ny*):	2
Translational error compensation data value (C*nz*):	3
Rotary error compensation data value (C$n\alpha$):	4
Rotary error compensation data value (C$n\beta$):	5
Rotary error compensation data value (C$n\gamma$):	-1

- Beginning and end of a record

A Three-dimensional rotary error compensation data record begins with % and ends with %.

Example (Tool head rotation type machine)

```
% ; .....Beginning of record
N200001 A1 P1 A2 P2 A3 P3 A4 P2 A5 P3 A6 P-1 ;      Compensation value of
N200002 A1 P0 A2 P0 A3 P-3 A4 P2 A5 P3 A6 P-1;      compensation space for linear axes
    :                                                  :
N207001 A1 P1 A2 P2 A3 P3 A4 P2 A5 P1 A6 P-1 ;      Compensation value of
N207002 A1 P1 A2 P2 A3 P3 A4 P2 A5 P3 A6 P-11 ;     compensation space for rotary axes
N207812 A1 P1 A2 P2 A3 P3 A4 P2 A15 P3 A6 P-1 ;        :
% .....End of record
```

- Input of compensation data using G10

Compensation data can be changed from a machining program, using the programmable parameter input function.

The command format is as follows:

```
%
G10 L53 ;
N_ P_ R_ ;
N_ P_ R_ ;
:
G11 ;
%
```

G10 L53: Three-dimensional rotary error compensation data input mode
G11 : Cancellation of three-dimensional rotary error compensation data input mode
N : Compensation point number (1-7812)
P1 : Translational error compensation value (C*nx*)
P2 : Translational error compensation value (C*ny*)
P3 : Translational error compensation value (C*nz*)
P4 : Rotary error compensation value (C$n\alpha$)
P5 : Rotary error compensation value (C$n\beta$)
P6 : Rotary error compensation value (C$n\gamma$)
R : Compensation data (-128-127)

NOTE
1 In three-dimensional rotary error compensation data input mode, no other NC statements can be commanded.
2 The decimal point cannot be used in address N, P, and R.

8.2.7 Inputting and Outputting Custom Macro Common Variables

8.2.7.1 Inputting custom macro common variables

The value of a custom macro common variable is loaded into the memory of the CNC from an external device. The same format used to output custom macro common variables is used for input.

Inputting custom macro common variables (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the continuous menu key  until soft key [MACRO] appears.
Press the soft key [MACRO].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key  until soft key [F INPUT] appears.
Press the soft key [F INPUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "MACRO.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the custom macro common variables, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the custom macro common variables, press the soft key [CAN].

Inputting custom macro common variables (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [MACRO] appears.
Press the vertical soft key [MACRO].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the horizontal soft key [F INPUT].
6 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "MACRO.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts reading the custom macro common variables, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the custom macro common variables, press the horizontal soft key [CAN].

Explanation

- Common variables

The common variables (#500 to #549) can be input and output.
(When the option for addition of common variable is specified, values from #500 to #999 can be input and output.) #100 to #149 can be input when bit 3 (PV5) of parameter No. 6001 is set to 1. (When the option for addition of common variable is specified, values from #100 to #199 can be input and output.)

8.2.7.2 Outputting custom macro common variables

Custom macro common variables stored in the memory of the CNC can be output in a defined output format to an external device.

Outputting custom macro common variables (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the continuous menu key  until soft key [MACRO] appears.
Press the soft key [MACRO].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key  until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name "MACRO.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the custom macro common variables, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting custom macro common variables (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .

3 Press the vertical soft key [NEXT PAGE] until vertical soft key [MACRO] appears.
Press the vertical soft key [MACRO].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the horizontal soft key [F OUTPUT].
6 Type the file name that you want to output.
If the file name is omitted, default file name "MACRO.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the custom macro common variables, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Explanation

- Output format

The output format is as follows:
The values of custom macro variables are output in a bit-image hexadecimal representation of double-precision floating-point type data.

```
%
G10L85P100(0000000000000000)
G10L85P101(0000000000000000)
G10L85P102(FFFFFFFFFFFFFFFF)
 :
G10L85P500(4024000000000000)
G10L85P501(4021000000000000)
G10L85P502(0000000000000000)
 .
SETVN500[ABC]
SETVN501[DEF]
SETVN502[GHI]
 :
M02
%

    P_ : Macro variable number
    SETVN_ [ _ ] : Common variable name
```

NOTE
The conventional custom macro statement program format cannot be used for output.

By setting bit 0 (MCO) of parameter No. 6019, it is possible to output macro variable numbers and variable data values as comments after normally output data.
The output comments do not affect data input.

- Common variable

The common variables (#500 to #549) can be input and output.
(When the option for addition of common variable is specified, values from #500 to #999 can be input and output.) #100 to #149 can be output when bit 3 (PV5) of parameter No. 6001 is set to 1. (When the option for addition of common variable is specified, values from #100 to #199 can be input and output.)

8.2.8 Inputting and Outputting Workpiece Coordinates System Data

8.2.8.1 Inputting workpiece coordinate system data

Coordinate system variable data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When coordinate system variable data with a data number corresponding to existing coordinate system variable data registered in the memory is loaded, the loaded coordinate system variable data replaces the existing coordinate system variable data.

Inputting workpiece coordinate system data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the continuous menu key until soft key [WORK] appears.
Press the soft key [WORK].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key until soft key [F INPUT] appears.
Press the soft key [F INPUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "EXT_WKZ.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the workpiece coordinate system data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the workpiece coordinate system data, press the soft key [CAN].

Inputting workpiece coordinate system data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [WORK] appears.
Press vertical soft key [WORK].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the horizontal soft key [F INPUT].
6 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "EXT_WKZ.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts reading the workpiece coordinate system data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the workpiece coordinate system data, press the horizontal soft key [CAN].

8.2.8.2 Outputting workpiece coordinate system data

All coordinate system variable data is output in the output format from the memory of the CNC to an external device.

Outputting workpiece coordinate system data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the continuous menu key ▷ until soft key [WORK] appears.
Press the soft key [WORK].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key ▷ until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name "EXT_WKZ.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the workpiece coordinate system data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting workpiece coordinate system data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [WORK] appears.
Press vertical soft key [WORK].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the horizontal soft key [F OUTPUT].
6 Type the file name that you want to output.
If the file name is omitted, default file name "EXT_WKZ.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the workpiece coordinate system data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.9 Inputting and Outputting Operation History Data

Only output operation is permitted on operation history data.
The output data is in text format. So, to reference the output data you must use an application that can handle text files on the personal computer.

8.2.9.1 Outputting operation history data

All operation history data is output in the output format form the memory of the CNC to an external device.

Outputting operation history data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key SYSTEM.
3 Press the continuous menu key ▷ until soft key [OPERAT HISTRY] appears.
Press the soft key [OPERAT HISTRY].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name "OPRT_HIS.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the operation history data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting operation history data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key SYSTEM.
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [OPERAT HISTRY] appears.
Press the vertical soft key [OPERAT HISTRY].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the horizontal soft key [F OUTPUT].
6 Type the file name that you want to output.
If the file name is omitted, default file name "OPRT_HIS.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the operation history data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.9.2 Inputting operation history signal selection data

Operation history signal selection data is input from an external input/output device to the memory of the CNC. The input format is the same as the format in which operation history signal selection data is output.

Inputting operation history signal section data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key SYSTEM.
3 Press the continuous menu key ▷ until soft key [OPERAT HISTRY] appears.
Press the soft key [OPERAT HISTRY].

4 Press the soft key [SIGNAL SELECT].
5 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
6 Press the soft key [(OPRT)].
7 Press the soft key [F INPUT].
8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "OHIS_SIG.TXT" is assumed.
9 Press the soft key [EXEC].
This starts reading the operation history signal selection data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the operation history signal selection data, press the soft key [CAN].

Inputting operation history signal section data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [OPERAT HISTRY] appears.
Press the vertical soft key [OPERAT HISTRY].
4 Press the vertical soft key [SIGNAL SELECT].
5 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
6 Press the horizontal soft key [F INPUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "OHIS_SIG.TXT" is assumed.
8 Press the horizontal soft key [EXEC].
This starts reading the operation history signal selection data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the operation history signal selection data, press the horizontal soft key [CAN].

8.2.9.3 Outputting operation history signal section data

Operation history signal selection data is output in the output format form the memory of the CNC to an external device.

Outputting operation history signal section data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the continuous menu key  until soft key [OPERAT HISTRY] appears.
Press the soft key [OPERAT HISTRY].
4 Press the soft key [SIGNAL SELECT].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
6 Press the soft key [(OPRT)].
7 Press the soft key [F OUTPUT].
8 Type the file name that you want to output.
If the file name is omitted, default file name "OHIS_SIG.TXT" is assumed.
9 Press the soft key [EXEC].
This starts outputting the operation history signal selection data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting operation history signal section data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [OPERAT HISTRY] appears.
Press the vertical soft key [OPERAT HISTRY].
4 Press the vertical soft key [SIGNAL SELECT].
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
6 Press the horizontal soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name "OHIS_SIG.TXT" is assumed.
8 Press the horizontal soft key [EXEC].
This starts outputting the operation history signal selection data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.9.4 Input/output format of operation history signal selection data

Operation history signal selection data is input and output in the following input and output formats.

- Keywords

The following alphabets are used as keywords.
The numeric value following each keyword has the meaning listed below:

Table 8.2.9.4 (a)

Keyword	Meaning of the following numeric value
N	Data number
L	PMC number
X,Y,G,F,P	Signal address
Q	Bit state

- Format

Operation history signal selection data is output in the following format:

N	**	L	**	X,Y,G,F,P	*****	Q	********	;

The numeric value following the N represents an operation history signal selection data number.
The numeric value following the L represents a PMC number in a multipath PMC.
The numeric value following the X, Y, G, and F represents the signal address to be recorded in operation history. Here, P is a special keyword indicating that none is set.
The numeric value following the Q represents the bit position of the signal to be recorded in operation history.
The semicolon (;) indicates the end of block (LF in the ISO code or CR in the EIA code).

Example 1
N01 L01 G00043 Q00100111;
As operation history signal selection data number 1, bits 0, 1, 2, and 5 of G0043 of the first PMC are set.

Example 2
N02 L00 P00000 Q00000000;
For operation history signal selection data number 2, no signal selection data is set.

- Beginning and end of a record

A operation history signal selection data record begins with % and ends with %.

Example
```
% ; .....Beginning of record
N01 L01 G00043 Q00100111;
N02 L01 F00001 Q00000001;
 :
N60 L00 P00000 Q00000000;
% .....End of record
```

8.2.10 Inputting and Outputting Tool Management Data

NOTE
1 For multi-path systems, place all paths in the EDIT mode before performing input and output operations.
2 The format used is the same as the registration format of the G10 format.

8.2.10.1 Inputting tool management data

Tool management data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When tool management data with a data number corresponding to existing tool management data registered in the memory is loaded, the loaded tool management data replaces the existing tool management data.

Inputting tool management data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key  to display the tool management screen or magazine screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F INPUT].
6 Press the soft key [TOOL].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "TOOL_MNG.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the tool management data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the tool management data, press the soft key [CAN].

NOTE
1 When using oversize tool support of the tool management function, keep the following in mind.
   - If a target tool is registered in a cartridge and interferes with other tools in registration or modification of tool geometry data of the tool management data, PS alarm 5360 is issued. (The data is not input.)
   - When restoring backup data in the state where all data related to the tool management function in the NC is cleared, restore tool geometry data, tool management data, and cartridge management table data in this order.
2 When bit 4 (OVI) of parameter No. 13206 to 1, tool management data, tool offset data, and tool geometry size data can be input as one file.

Inputting tool management data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears. Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE] or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F INPUT].
7 Press the horizontal soft key [TOOL].
8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "TOOL_MNG.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the tool management data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the tool management data, press the horizontal soft key [CAN].

NOTE
1 When using oversize tool support of the tool management function, keep the following in mind.
   - If a target tool is registered in a cartridge and interferes with other tools in registration or modification of tool geometry data of the tool management data, PS alarm 5360 is issued. (The data is not input.)
   - When restoring backup data in the state where all data related to the tool management function in the NC is cleared, restore tool geometry data, tool management data, and cartridge management table data in this order.
2 When bit 4 (OVI) of parameter No. 13206 to 1, tool management data, tool offset data, and tool geometry size data can be input as one file.

8.2.10.2 Outputting tool management data

All tool management data is output in the output format from the memory of the CNC to an external device.

Outputting tool management data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.

2 Press the function key  to display the tool management screen or magazine screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT].
6 Press the soft key [TOOL].
7 Type the file name that you want to output.
If the output file name is omitted, default output file name "TOOL_MNG.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the tool management data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

NOTE
When bit 4 (OVI) of parameter No. 13206 to 1, tool management data, tool offset data, and tool geometry size data can be output as one file. The format of tool offset data output using this function cannot be used with the programmable data input function.

Outputting tool management data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE] or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F OUTPUT].
7 Press the horizontal soft key [TOOL].
8 Type the file name that you want to output.
If the output file name is omitted, default output file name "TOOL_MNG.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts outputting the tool management data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

NOTE
When bit 4 (OVI) of parameter No. 13206 to 1, tool management data, tool offset data, and tool geometry size data can be output as one file. The format of tool offset data output using this function cannot be used with the programmable data input function.

8.2.10.3 Inputting magazine data

Magazine data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When magazine data with a data number corresponding to existing magazine data registered in the memory is loaded, the loaded magazine data replaces the existing magazine data.

Inputting magazine data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.

2 Press the function key  to display the tool management screen or magazine screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F INPUT].
6 Press the soft key [MAGAZINE].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "MAGAZINE.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the magazine data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the magazine data, press the soft key [CAN].

NOTE
When using oversize tool support of the tool management function, keep the following in mind.
- If a tool interferes with other tools in registration in or modification to the cartridge management table, PS alarm 5360 is issued. (The data is not input.)
- When restoring backup data in the state where all data related to the tool management function in the NC is cleared, restore tool geometry data, tool management data, and cartridge management table data in this order.

Inputting magazine data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE] or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F INPUT].
7 Press the horizontal soft key [MAGAZINE].
8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "MAGAZINE.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the magazine data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the magazine data, press the horizontal soft key [CAN].

NOTE
When using oversize tool support of the tool management function, keep the following in mind.
- If a tool interferes with other tools in registration in or modification to the cartridge management table, PS alarm 5360 is issued. (The data is not input.)
- When restoring backup data in the state where all data related to the tool management function in the NC is cleared, restore tool geometry data, tool management data, and cartridge management table data in this order.

8.2.10.4 Outputting magazine data

All magazine data is output in the output format from the memory of the CNC to an external device.

Outputting magazine data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key  to display the tool management screen or magazine screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT].
6 Press the soft key [MAGAZINE].
7 Type the file name that you want to output.
If the file name is omitted, default file name "MAGAZINE.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the magazine data, and "OUTPUT" blinks in the lower right part of the screen.
When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting magazine data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE] or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F OUTPUT].
7 Press the horizontal soft key [MAGAZINE].
8 Type the file name that you want to output.
If the file name is omitted, default file name "MAGAZINE.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts outputting the magazine data, and "OUTPUT" blinks in the lower right part of the screen.
When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.10.5 Inputting tool life status name data

Tool life status name data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When tool life status name data with a data number corresponding to existing tool life status name data registered in the memory is loaded, the loaded tool life status name data replaces the existing tool life status name data.

Inputting tool life status name data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key  to display the tool management screen or magazine screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].

5 Press the soft key [F INPUT].
6 Press the soft key [STATUS].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “STATUS.TXT” is assumed.
8 Press the soft key [EXEC].
This starts reading the tool life status name data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the tool life status name data, press the soft key [CAN].

Inputting tool life status name data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE] or [TOOL].
5 Press the EDIT switch on the machine operator’s panel.
6 Press the horizontal soft key [F INPUT].
7 Press the horizontal soft key [STATUS].
8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “STATUS.TXT” is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the tool life status name data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the tool life status name data, press the horizontal soft key [CAN].

8.2.10.6 Outputting tool life status name data

All tool life status name data is output in the output format from the memory of the CNC to an external device.

Outputting tool life status name data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key  to display the tool management screen or magazine screen.
3 Press the EDIT switch on the machine operator’s panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT].
6 Press the soft key [STATUS].
7 Type the file name that you want to output.
If the file name is omitted, default file name “STATUS.TXT” is assumed.
8 Press the soft key [EXEC].
This starts outputting the tool life status name data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting tool life status name data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.

2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE] or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F OUTPUT].
7 Press the horizontal soft key [STATUS].
8 Type the file name that you want to output.
If the file name is omitted, default file name "STATUS.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts outputting the tool life status name data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.10.7 Inputting name data of customize data

Name data of customize data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When name data of customize data with a data number corresponding to existing name data of customize data registered in the memory is loaded, the loaded name data of customize data replaces the existing data.

Inputting name data of customize data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key  to display the tool management screen or magazine screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F INPUT].
6 Press the soft key [CUSTOM].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "CUSTOMIZ.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the name data of customize data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the name data of customize data, press the soft key [CAN].

Inputting name data of customize data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE] or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F INPUT].
7 Press the horizontal soft key [CUSTOM].
8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "CUSTOMIZ.TXT" is assumed.

9 Press the horizontal soft key [EXEC].
This starts reading the name data of customize data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the name data of customize data, press the horizontal soft key [CAN].

8.2.10.8 Outputting name data of customize data

All name data of customize data is output in the output format from the memory of the CNC to an external device.

Outputting name data of customize data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key  to display the tool management screen or magazine screen.
3 Press the EDIT switch on the machine operator’s panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT].
6 Press the soft key [CUSTOM].
7 Type the file name that you want to output.
If the file name is omitted, default file name “CUSTOMIZ.TXT” is assumed.
8 Press the soft key [EXEC].
This starts outputting the name data of customize data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting name data of customize data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE] or [TOOL].
5 Press the EDIT switch on the machine operator’s panel.
6 Press the horizontal soft key [F OUTPUT].
7 Press the horizontal soft key [CUSTOM].
8 Type the file name that you want to output.
If the file name is omitted, default file name “CUSTOMIZ.TXT” is assumed.
9 Press the horizontal soft key [EXEC].
This starts outputting the name data of customize data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.10.9 Inputting customize data displayed as tool management data

Customize data displayed as tool management data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When customize data displayed as tool management data with a data number corresponding to existing customize data displayed as tool management data registered in the memory is loaded, the loaded customize data replaces the existing data.

Inputting customize data displayed as tool management data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key  to display the tool management screen, magazine screen, or each tool data screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F INPUT].
6 Press the soft key [DSP-LAYOUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "DISPCSTM.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the customize data displayed as tool management data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the customize data displayed as tool management data, press the soft key [CAN].

Inputting customize data displayed as tool management data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE], [EACH TOOL], or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F INPUT].
7 Press the horizontal soft key [DSP-LAYOUT].
8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "DISPCSTM.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the customize data displayed as tool management data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the customize data displayed as tool management data, press the horizontal soft key [CAN].

8.2.10.10 Outputting customize data displayed as tool management data

Customize data displayed as tool management data is output from the memory of the CNC to an external device in the output format.

Outputting customize data displayed as tool management data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.

2 Press the function key  to display the tool management screen, magazine screen, or each tool data screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT].
6 Press the soft key [DSP-LAYOUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name "DISPCSTM.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the customize data displayed as tool management data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting customize data displayed as tool management data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE], [EACH TOOL], or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F OUTPUT].
7 Press the horizontal soft key [DSP-LAYOUT].
8 Type the file name that you want to output.
If the file name is omitted, default file name "DISPCSTM.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts outputting the customize data displayed as tool management data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.10.11 Inputting spindle waiting position name data

Spindle waiting position name data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When spindle waiting position name data with a data number corresponding to existing spindle waiting position name data registered in the memory is loaded, the loaded spindle waiting position name data replaces the existing data.

Inputting spindle waiting position name data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key  to display the tool management screen, magazine screen, or each tool data screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F INPUT].
6 Press the soft key [SPWP-NAME].

7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “POSNAME.TXT” is assumed.
8 Press the soft key [EXEC].
This starts reading the spindle waiting position name data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the spindle waiting position name data, press the soft key [CAN].

Inputting spindle waiting position name data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE], [EACH TOOL], or [TOOL].
5 Press the EDIT switch on the machine operator’s panel.
6 Press the horizontal soft key [F INPUT].
7 Press the horizontal soft key [SPWP-NAME].
8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “POSNAME.TXT” is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the spindle waiting position name data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the spindle waiting position name data, press the horizontal soft key [CAN].

8.2.10.12 Outputting spindle waiting position name data

Spindle waiting position name data is output from the memory of the CNC to an external device in the output format.

Outputting spindle waiting position name data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key  to display the tool management screen, magazine screen, or each tool data screen.
3 Press the EDIT switch on the machine operator’s panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT].
6 Press the soft key [SPWP-NAME].
7 Type the file name that you want to output.
If the file name is omitted, default file name “POSNAME.TXT” is assumed.
8 Press the soft key [EXEC].
This starts outputting the spindle waiting position name data display, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting spindle waiting position name data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .

3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears. Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE], [EACH TOOL], or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F OUTPUT].
7 Press the horizontal soft key [SPWP-NAME].
8 Type the file name that you want to output.
If the file name is omitted, default file name "POSNAME.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts outputting the spindle waiting position name data display, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.10.13 Inputting decimal point position data of customize data

Decimal point position data of customize data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When decimal point position data of customize data with a data number corresponding to existing decimal point position data of customize data registered in the memory is loaded, the loaded decimal point position data of customize data replaces the existing data.

Inputting decimal point position data of customize data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key 

to display the tool management screen, magazine screen, or each tool data screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F INPUT].
6 Press the soft key [CSTM-DPOINT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "POINTPOS.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the decimal point position data of customize data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the decimal point position data of customize data, press the soft key [CAN].

Inputting decimal point position data of customize data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key 

.
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears. Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE], [EACH TOOL], or [TOOL].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F INPUT].
7 Press the horizontal soft key [CSTM-DPOINT].
8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "POINTPOS.TXT" is assumed.

9 Press the horizontal soft key [EXEC].
This starts reading the decimal point position data of customize data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the decimal point position data of customize data, press the horizontal soft key [CAN].

8.2.10.14 Outputting decimal point position data of customize data

Decimal point position data of customize data is output from the memory of the CNC to an external device in the output format.

Outputting decimal point position data of customize data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key 

to display the tool management screen, magazine screen, or each tool data screen.
3 Press the EDIT switch on the machine operator’s panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT].
6 Press the soft key [CSTM-DPOINT].
7 Type the file name that you want to output.
If the file name is omitted, default file name “POINTPOS.TXT” is assumed.
8 Press the soft key [EXEC].
This starts outputting the decimal point position data of customize data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting decimal point position data of customize data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key 

.
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE], [EACH TOOL], or [TOOL].
5 Press the EDIT switch on the machine operator’s panel.
6 Press the horizontal soft key [F OUTPUT].
7 Press the horizontal soft key [CSTM-DPOINT].
8 Type the file name that you want to output.
If the file name is omitted, default file name “POINTPOS.TXT” is assumed.
9 Press the horizontal soft key [EXEC].
This starts outputting the decimal point position data of customize data, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.10.15 Inputting tool geometry data

Tool geometry data is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When tool geometry data with a data number corresponding to existing tool geometry data registered in the memory is loaded, the loaded tool geometry data replaces the existing data.

Inputting tool geometry data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key  to display the tool management screen, magazine screen, each tool data screen, or tool geometry data screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F INPUT].
6 Press the soft key [TOOL- GEOM].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "TOOLGEOM.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the tool geometry data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the tool geometry data, press the soft key [CAN].

NOTE
1 If the tool with a number of tool geometry data to be changed is registered to the magazine when an attempt is made to change the tool geometry data, an alarm PS5360 is issued. (The data is not input.)
2 After data related to the tool management functions in the NC has all been cleared, restore backup data in the following order: Tool geometry data, tool management data, and magazine management table.

Inputting tool geometry data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE], [EACH TOOL], [TOOL], or [TOOL- GEOM].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F INPUT].
7 Press the horizontal soft key [TOOL- GEOM].
8 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "TOOLGEOM.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the tool geometry data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the tool geometry data, press the horizontal soft key [CAN].

NOTE
1 If the tool with a number of tool geometry data to be changed is registered to the magazine when an attempt is made to change the tool geometry data, an alarm PS5360 is issued. (The data is not input.)
2 After data related to the tool management functions in the NC has all been cleared, restore backup data in the following order: Tool geometry data, tool management data, and magazine management table.

8.2.10.16 Outputting tool geometry data

Tool geometry data is output from the memory of the CNC to an external device in the output format.

Outputting tool geometry data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key  to display the tool management screen, magazine screen, each tool data screen, or tool geometry data screen.
3 Press the EDIT switch on the machine operator's panel.
4 Press the soft key [(OPRT)].
5 Press the soft key [F OUTPUT].
6 Press the soft key [TOOL- GEOM].
7 Type the file name that you want to output.
If the output file name is omitted, default output file name "TOOLGEOM.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the tool geometry data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting tool geometry data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL MANAGER] appears.
Press the vertical soft key [TOOL MANAGER].
4 Press the vertical soft key [MAGAZINE], [EACH TOOL], [TOOL], or [TOOL- GEOM].
5 Press the EDIT switch on the machine operator's panel.
6 Press the horizontal soft key [F OUTPUT].
7 Press the horizontal soft key [TOOL- GEOM].
8 Type the file name that you want to output.
If the output file name is omitted, default output file name "TOOLGEOM.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts outputting the tool geometry data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.11 Inputting and Outputting Workpiece Setting Error Compensation Value

8.2.11.1 Inputting values on the workpiece setting error compensation screen

Workpiece setting error compensation value is loaded into the memory of the CNC from an external device. The input format is the same as the output format. When tool management data with a data number corresponding to existing workpiece setting error compensation value registered in the memory is loaded, the loaded tool management data replaces the existing workpiece setting error compensation value.

Inputting workpiece setting error compensation value (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the continuous menu key  until soft key [WORK SET ER] appears.
Press the soft key [WORK SET ER]. Workpiece setting error compensation screen appears.
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key  until soft key [F INPUT] appears.
Press the soft key [F INPUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "WSEC_VAL.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the workpiece setting error compensation value, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the workpiece setting error compensation value, press the soft key [CAN].

<Default file name>
When bit 0 (MDP) of parameter No. 138 is set to 1, a path number is added as the extension of a default file name.

Table 8.2.11 (a) Workpiece setting error file name

Selected path number	Bit 0 of parameter No. 0138	
	0	1
1	WSEC_VAL.TXT	WSEC_VAL.TXT
2	WSEC_VAL.TXT	WSEC_VAL.P-2
3	WSEC_VAL.TXT	WSEC_VAL.P-3
4	WSEC_VAL.TXT	WSEC_VAL.P-4
:		
10	WSEC_VAL.TXT	WSEC_VAL.P10

Inputting workpiece setting error compensation value (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [WORK SET ER] appears.
Press the vertical soft key [WORK SET ER]. Workpiece setting error compensation screen appears.
4 Press the EDIT switch on the machine operator's panel or enter state emergency stop.

5 Press the horizontal soft key [F INPUT].
6 Type the name of the file that you want to input.
If the input file name is omitted, default input file name “WSEC_VAL.TXT” is assumed.
7 Press the horizontal soft key [EXEC].
This starts reading the workpiece setting error compensation value, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the workpiece setting error compensation value, press the horizontal soft key [CAN].

8.2.11.2 Outputting values on the workpiece setting error compensation screen

Workpiece setting error compensation value is output in a defined output format from the memory of the CNC to an external device.

Outputting workpiece setting error compensation value (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the continuous menu key until soft key [WORK SET ER] appears.
Press the soft key [WORK SET ER]. Workpiece setting error compensation screen appears.
4 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
5 Press the soft key [(OPRT)].
6 Press the continuous menu key until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the file name is omitted, default file name “WSEC_VAL.TXT” is assumed.
8 Press the soft key [EXEC].
This starts writing the workpiece setting error compensation value, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting workpiece setting error compensation value (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the function key .
3 Press the vertical soft key [NEXT PAGE] until vertical soft key [WORK SET ER] appears.
Press the vertical soft key [WORK SET ER]. Workpiece setting error compensation screen appears.
4 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
5 Press the horizontal soft key [F OUTPUT].
6 Type the file name that you want to output.
If the file name is omitted, default file name “WSEC_VAL.TXT” is assumed.
7 Press the horizontal soft key [EXEC].
This starts writing the workpiece setting error compensation value, and “OUTPUT” blinks in the lower right part of the screen. When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.2.11.3 Input/output format of workpiece setting error values

Workpiece setting error values are input and output in the following format.

```
%
G10L23P0 X_ Y_ Z_ (I_ J_ )
G10L23P1 X_ Y_ Z_ A_ B_ C_ (I_ J_ )
G10L23P2 X_ Y_ Z_ A_ B_ C_ (I_ J_ )
G10L23P3 X_ Y_ Z_ A_ B_ C_ (I_ J_ )
G10L23P4 X_ Y_ Z_ A_ B_ C_ (I_ J_ )
G10L23P5 X_ Y_ Z_ A_ B_ C_ (I_ J_ )
G10L23P6 X_ Y_ Z_ A_ B_ C_ (I_ J_ )
G10L23P7 X_ Y_ Z_ A_ B_ C_ (I_ J_ )
(M02)
%
```

P: Workpiece setting error number 0 to 7
X: X-direction error Δx
Y: Y-direction error Δx
Z: Z-direction error Δx
A: Rotational error Δa
B: Rotational error Δb
C: Rotational error Δc
I: Table rotation axis position 1
J: Table rotation axis position 2

NOTE

1 If an address other than P is omitted, the amount of workpiece setting error corresponding to the omitted address is assumed to be the amount before specification.
2 The specified value is used as the amount of workpiece setting error as is. Even when a value is specified in the G91 mode, it is used as the amount of workpiece setting error as is. The specified value is not added to the value before specification.
3 For workpiece setting error number 0 (COMMON), any rotational error cannot be set. If rotational error A, B, or C is specified for setting error number 0, alarm PS1144, "G10 FORMAT ERROR", is issued.
4 When there is only one table rotation axis, table rotation axis position 2 cannot be set. When there is no table rotation axis, both table rotation axis positions 1 and 2 cannot be set. If an attempt is made to set the position, alarm PS1144, “G10 FORMAT ERROR", is issued.
5 If an address other than P, X, Y, Z, A, B, C, I, or J is specified, alarm PS1144, “G10 FORMAT ERROR”, is issued.
6 If a value other than 0 to 7 is specified for workpiece setting error number P, alarm PS0031, “ILLEGAL P COMMAND IN G10”, is issued.
7 An NC address with no decimal point is assumed as follows depending on the setting of bit 0 (DPI) of parameter No. 3401:
0 : Least input increment
1 : Unit of mm or inches

8.2.12 Inputting and Outputting Tool Life Management Data

8.2.12.1 Inputting tool life management data

Tool life management data is loaded into the memory of the CNC from an external device such as a memory card.

Inputting tool life management data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the EDIT switch on the machine operator's panel.
3 Press the function key .
4 Press the continuous menu key  and chapter selection soft key [TOOL LIFE].
5 Press the soft key [(OPRT)].
When the tool life management (group editing) screen is displayed, press soft key [END] to display the tool life management (list) screen.
6 Press the soft key [F INPUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "TOOLLIFE.TXT" is assumed.
8 Press the soft key [EXEC].
This starts reading the tool life management data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

NOTE
Before tool life management data is input, all tool life management data is erased.

Inputting tool life management data (for 15/19-inch display unit)

Procedure

1 Make sure the input device is ready for reading.
2 Press the EDIT switch on the machine operator's panel.
3 Press the function key .
4 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL LIFE] appears.
5 Press the vertical soft key [TOOL LIFE].
When the tool life management (group editing) screen is displayed, press soft key [END] to display the tool life management (list) screen.
6 Press the horizontal soft key [F INPUT].
7 Type the name of the file that you want to input.
If the input file name is omitted, default input file name "TOOLLIFE.TXT" is assumed.
8 Press the horizontal soft key [EXEC].
This starts reading the tool life management data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

8.2.12.2 Outputting tool life management data

Tool life management data stored in the memory of the CNC is output to an external device such as a memory card.

Outputting tool life management data (for 8.4/10.4-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator's panel.
3 Press the function key .
4 Press the continuous menu key  and chapter selection soft key [TOOL LIFE].
5 Press the soft key [(OPRT)].
When the tool life management (group editing) screen is displayed, press soft key [END] to display the tool life management (list) screen.
6 Press the soft key [F OUTPUT].
7 Type the file name that you want to output.
If the output file name is omitted, default output file name "TOOLLIFE.TXT" is assumed.
8 Press the soft key [EXEC].
This starts outputting the tool life management data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting tool life management data (for 15/19-inch display unit)

Procedure

1 Make sure the output device is ready for output.
2 Press the EDIT switch on the machine operator's panel.
3 Press the function key .
4 Press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL LIFE] appears.
5 Press the vertical soft key [TOOL LIFE].
When the tool life management (group editing) screen is displayed, press soft key [END] to display the tool life management (list) screen.
6 Press the horizontal soft key [F OUTPUT].
8 Type the file name that you want to output.
If the output file name is omitted, default output file name "TOOLLIFE.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts outputting the tool life management data, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3 INPUT/OUTPUT ON THE ALL IO SCREEN

Just by using the ALL IO screen, you can input and output programs, parameters, offset data, pitch error compensation data, macro variables, workpiece coordinate system data, operation history data, and tool management data to an external I/O device.

⚠ CAUTION
While an external I/O device is being accessed, do not turn the power to the CNC off or remove the device, or the device may be damaged.

NOTE
The ALL IO screen can be operated only if a memory card or USB memory is selected as an external I/O device. Use NC parameter No. 20 to select an external I/O device.

The following explains how to display the ALL IO screen:

Displaying the ALL IO screen (for 8.4/10.4-inch display unit)

Procedure

1 Press the function key .
2 Press the continuous menu key  several times.
3 Press the soft key [ALL IO] to display the ALL IO screen.

The subsequent steps to select data from the ALL IO screen will be explained for each type of data.

Displaying the ALL IO screen (for 15/19-inch display unit)

Procedure

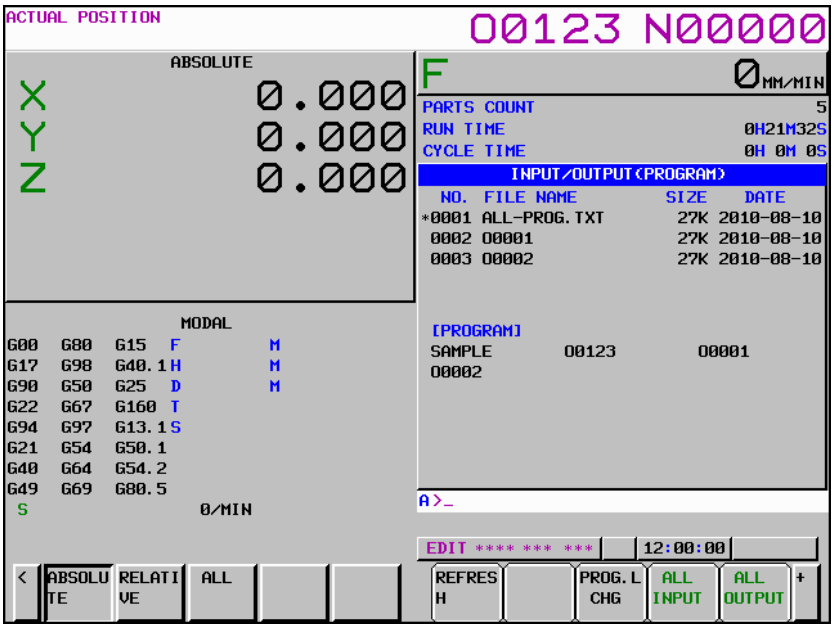
1 Press the function key .
2 Press the vertical soft key [NEXT PAGE] until vertical soft key [ALL I/O] appears.
3 Press the vertical soft key [ALL I/O] to display the ALL IO screen.

The subsequent steps to select data from the ALL IO screen will be explained for each type of data.

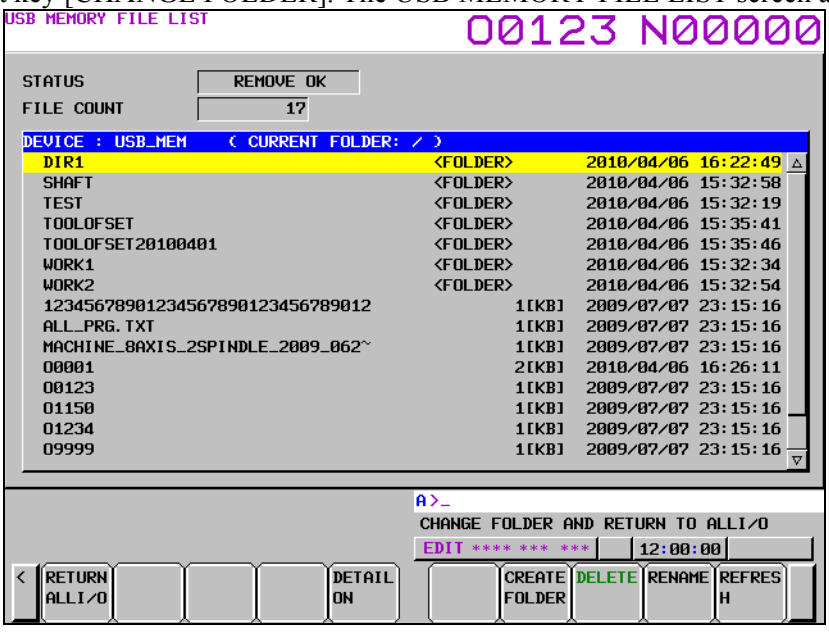
Procedure for changing the target folder for inputting and outputting data (for 8.4/10.4-inch display unit)

The last folder displayed on the USB MEMORY FILE LIST screen is treated as the default target folder for inputting and outputting data in a USB memory. If you want to input data from or output it to another folder, perform the following operation.

1 Display the ALL IO screen.



2 Press the soft key [CHANGE FOLDER]. The USB MEMORY FILE LIST screen appears.



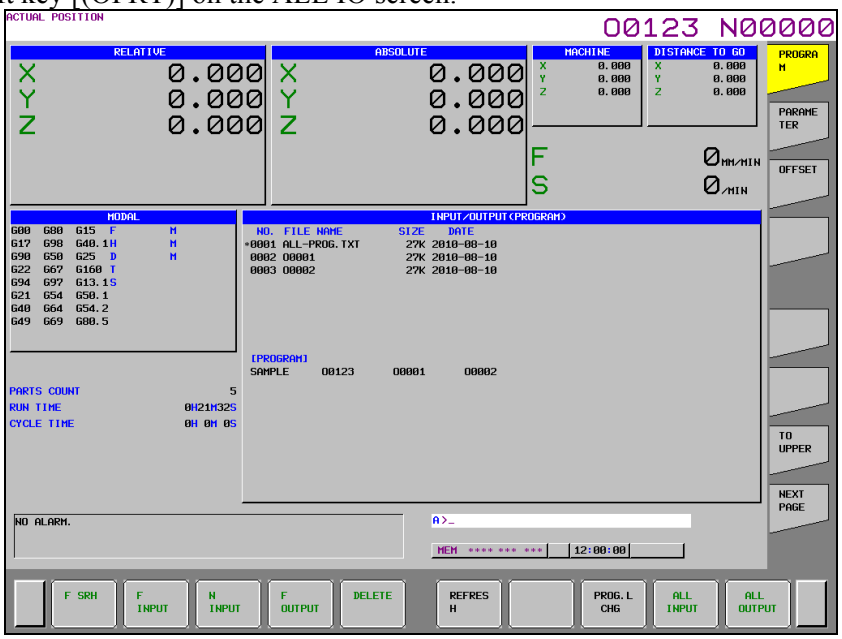
3 Display a folder you want to use for inputting and outputting data.

4 Press the soft key [RETURN ALL I/O]. The ALL IO screen appears again. The folder displayed on the list screen is used as the target folder for inputting and outputting data.

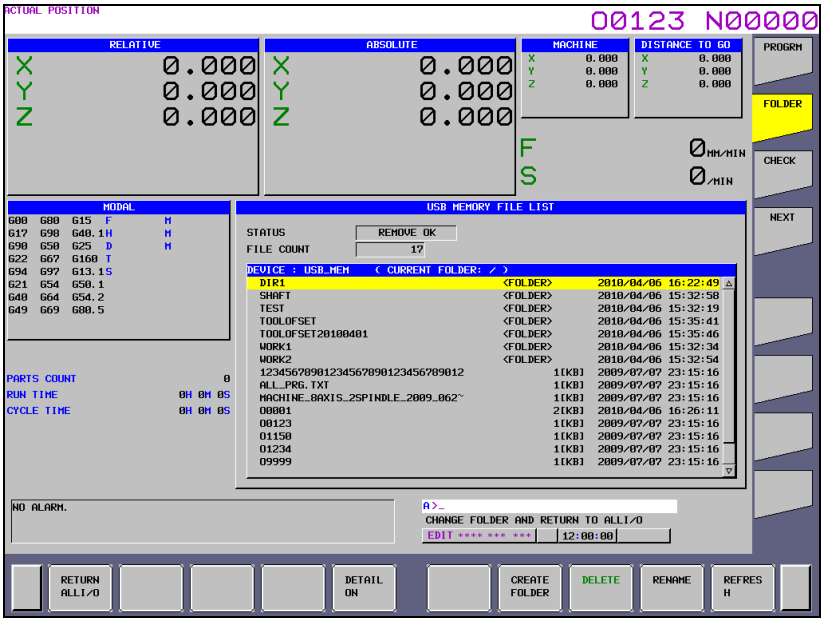
Procedure for changing the target folder for inputting and outputting data (for 15/19-inch display unit)

The last folder displayed on the USB MEMORY FILE LIST screen is treated as the default target folder for inputting and outputting data in a USB memory. If you want to input data from or output it to another folder, perform the following operation.

1 Press the soft key [(OPRT)] on the ALL IO screen.



2 Press the horizontal soft key [CHANGE FOLDER]. The USB MEMORY FILE LIST screen appears.



3 Display a folder you want to use for inputting and outputting data.

4 Press the soft key [RETURN ALL I/O]. The ALL IO screen appears again. The folder displayed on the list screen is used as the target folder for inputting and outputting data.

Configuration of this section

Section 8.3, "INPUT/OUTPUT ON THE ALL IO SCREEN", consists of the following subsections:

8.3.1 Inputting/Outputting a Program 1227
8.3.2 Inputting and Outputting all Programs and Folders 1229
8.3.3 Inputting and Outputting Parameters 1231
8.3.4 Inputting and Outputting Offset Data 1232
8.3.5 Inputting/Outputting Pitch Error Compensation Data 1234
8.3.6 Inputting/Outputting Custom Macro Common Variables 1235
8.3.7 Inputting and Outputting Workpiece Coordinates System Data 1237
8.3.8 Inputting and Outputting Operation History Data 1238
8.3.9 Inputting and Outputting Tool Management Data 1239
8.3.10 Inputting and Outputting All Tool Management Data at a Time 1244
8.3.11 Inputting and Outputting Workpiece Setting Error Compensation Value 1247
8.3.12 File Format and Error Messages 1248

8.3.1 Inputting/Outputting a Program

A program can be input and output using the ALL IO screen.

Inputting a program (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [PROGRAM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "ALL-PROG.TXT" is assumed.
See the Table 8.3.1 (a) for details.
6 Set the program number to be used after the input.
Type a program number, and press the soft key [P SET].
If no program number is specified here, the program number in the file is adopted as is.
7 Press the soft key [EXEC].
This starts reading the program, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Table 8.3.1 (a)

[F SET]	[P SET]	Input file name	Input program	Input program name
BLANK	BLANK	ALL-PROG.TXT	All programs in ALL-PROG.TXT	File name at the time the file is saved
BLANK	INPUT	ALL-PROG.TXT	First program in ALL-PROG.TXT	Program name set with [P SET]
INPUT	BLANK	File name set with [F NAME]	All programs in the file specified with [F NAME]	File name at the time the file is saved
INPUT	INPUT	File name set with [F NAME]	First program in the file specified with [F NAME]	Program name set with [P SET]

Inputting a program (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [PROGRAM] appears.
Press the vertical soft key [PROGRAM].

2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the horizontal soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name "ALL-PROG.TXT" is assumed.
See the Table 8.3.1 (b) for details.
5 Set the program number to be used after the input.
Type a program number, and press the horizontal soft key [P SET].
If no program number is specified here, the program number in the file is adopted as is.
6 Press the horizontal soft key [EXEC].
This starts reading the program, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Table 8.3.1 (b)

[F SET]	[P SET]	Input file name	Input program	Input program name
BLANK	BLANK	ALL-PROG.TXT	All programs in ALL-PROG.TXT	File name at the time the file is saved
BLANK	INPUT	ALL-PROG.TXT	First program in ALL-PROG.TXT	Program name set with [P SET]
INPUT	BLANK	File name set with [F NAME]	All programs in the file specified with [F NAME]	File name at the time the file is saved
INPUT	INPUT	File name set with [F NAME]	First program in the file specified with [F NAME]	Program name set with [P SET]

Outputting a program (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [PROGRAM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the program that you want to output.
Type a program number, and press the soft key [P SET].
If O-9999 is typed, all programs in the memory are output.
6 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
When no file name is set, the output file name is assumed to be "O-number" if a single program number is specified; if O-9999 is specified, the output file name is assumed to be "ALL-PROG.TXT".
See the Table 8.3.1 (c) for details.
(For an explanation of foreground folders, see II-12.1.3, and for how to change foreground folders, see III-11.6.)
7 Press the soft key [EXEC].
This starts outputting the program, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Table 8.3.1 (c)

[F SET]	[P SET]	Output file name	Output program
BLANK	BLANK or (O-9999)	ALL-PROG.TXT	All programs in the foreground folders displayed in the program folder
BLANK	INPUT	Program name set with [P SET]	Program in the NC that is set with [P SET]
INPUT	BLANK or (O-9999)	File name set with [F NAME]	All programs in the foreground folders displayed in the program folder
INPUT	INPUT	File name set with [F NAME]	Program in the NC that is set with [P SET]

Outputting a program (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [PROGRAM] appears.
Press the vertical soft key [PROGRAM].
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the horizontal soft key [F OUTPUT].
4 Set the program that you want to output.
Type a program number, and press the horizontal soft key [P SET].
If O-9999 is typed, all programs in the memory are output.
5 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
When no file name is set, the output file name is assumed to be "O-number" if a single program number is specified; if O-9999 is specified, the output file name is assumed to be "ALL-PROG.TXT".
See the Table 8.3.1 (d) for details.
(For an explanation of foreground folders, see II-12.1.3, and for how to change foreground folders, see III-11.6.)
6 Press the horizontal soft key [EXEC].
This starts outputting the program, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Table 8.3.1 (d)

[F SET]	[P SET]	Output file name	Output program
BLANK	BLANK or (O-9999)	ALL-PROG.TXT	All programs in the foreground folders displayed in the program folder
BLANK	INPUT	Program name set with [P SET]	Program in the NC that is set with [P SET]
INPUT	BLANK or (O-9999)	File name set with [F NAME]	All programs in the foreground folders displayed in the program folder
INPUT	INPUT	File name set with [F NAME]	Program in the NC that is set with [P SET]

8.3.2 Inputting/Outputting all Programs and Folders

All programs and folders can be input and output using the ALL IO screen.

Inputting all programs and folders (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [PROGRAM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
Select EDIT mode or enter state emergency stop for all paths.
3 Press the soft key [(OPRT)].
4 Press the soft key [ALL INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "ALL-FLDR.TXT" is assumed.
7 Press the soft key [EXEC].
This starts reading the program, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting all programs and folders (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [PROGRAM] appears.
Press the vertical soft key [PROGRAM].
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
Select EDIT mode or enter state emergency stop for all paths.
3 Press the horizontal soft key [ALL INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name "ALL-FLDR.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the program, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Outputting all programs and folders (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [PROGRAM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [ALL OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "ALL-FLDR.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the program, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting all programs and folders (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [PROGRAM] appears.
Press the vertical soft key [PROGRAM].
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the horizontal soft key [ALL OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name "ALL-FLDR.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the program, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3.3 Inputting and Outputting Parameters

Parameters can be input and output using the ALL IO screen.

Inputting parameters (for 8.4/10.4-inch display unit)

Procedure

1 Press the function key .
2 Press the continuous menu key  until soft key [SETTING] appears.
Press the soft key [SETTING].
3 Press the MDI switch on the machine operator's panel or enter state emergency stop.
4 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data. Alarm SW0100 appears.
5 Press the soft key [PARAMETER] on the ALL IO screen.
6 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
7 Press the soft key [(OPRT)].
8 Press the soft key [N INPUT].
9 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "CNC-PARA.TXT" is assumed.
10 Press the soft key [EXEC].
This starts reading the parameter, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].
11 Press the function key .
12 Press the soft key [SETTING].
13 Press the MDI switch on the machine operator's panel or enter state emergency stop.
14 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
15 Turn the power of the CNC on again.

Inputting parameters (for 15/19-inch display unit)

Procedure

1 Press the function key .
2 Press the vertical soft key [NEXT PAGE] until vertical soft key [SETTING] appears.
Press the vertical soft key [SETTING].
3 Press the MDI switch on the machine operator's panel or enter state emergency stop.
4 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data. Alarm SW0100 appears.
5 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [PARAMETER] appears.
Press the vertical soft key [PARAMETER].
6 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
7 Press the horizontal soft key [N INPUT].
8 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name "CNC-PARA.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the parameter, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

10 Press the function key .
11 Press the vertical soft key [SETTING].
12 Press the MDI switch on the machine operator's panel or enter state emergency stop.
13 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
14 Turn the power of the CNC on again.

Outputting parameters (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [PARAMETER] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name "CNC-PARA.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the parameter, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting parameters (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [PARAMETER] appears.
Press the vertical soft key [PARAMETER].
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the file name is omitted, default file name "CNC-PARA.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the parameter, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3.4 Inputting and Outputting Offset Data

Offset data can be input and output using the ALL IO screen.

Inputting offset data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [OFFSET] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "TOOLOFST.TXT" is assumed.

6 Press the soft key [EXEC].
This starts reading the offset data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting offset data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [OFFSET] appears.
Press the vertical soft key [OFFSET].
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the horizontal soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name “TOOLOFST.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the offset data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Outputting offset data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [OFFSET] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name “TOOLOFST.TXT” is assumed.
6 Press the soft key [EXEC].
This starts outputting the offset data, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting offset data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [OFFSET] appears.
Press the vertical soft key [OFFSET].
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the file name is omitted, default file name “TOOLOFST.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the offset data, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3.5 Inputting/Outputting Pitch Error Compensation Data

Pitch error compensation data can be input and output using the ALL IO screen.

Inputting pitch error compensation data (for 8.4/10.4-inch display unit)

Procedure

1 Press the function key .
2 Press the continuous menu key  until soft key [SETTING] appears.
Press the soft key [SETTING].
3 Press the MDI switch on the machine operator's panel or enter state emergency stop.
4 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data. Alarm SW0100 appears.
5 Press the soft key [PITCH] on the ALL IO screen.
6 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
7 Press the soft key [(OPRT)].
8 Press the soft key [N INPUT].
9 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "PITCH.TXT" is assumed.
10 Press the soft key [EXEC].
This starts reading the pitch error compensation data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].
11 Press the function key .
12 Press the soft key [SETTING].
13 Press the MDI switch on the machine operator's panel or enter state emergency stop.
14 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
15 Turn the power of the CNC on again.

Inputting pitch error compensation data (for 15/19-inch display unit)

Procedure

1 Press the function key .
2 Press the vertical soft key [NEXT PAGE] until vertical soft key [SETTING] appears.
Press the vertical soft key [SETTING].
3 Press the MDI switch on the machine operator's panel or enter state emergency stop.
4 Enter 1 in response to the prompt for "PARAMETER WRITE" in setting data. Alarm SW0100 appears.
5 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [PITCH ERROR] appears.
Press the vertical soft key [PITCH ERROR].
6 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
7 Press the horizontal soft key [N INPUT].
8 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name "PITCH.TXT" is assumed.
9 Press the horizontal soft key [EXEC].
This starts reading the pitch error compensation data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

10 Press the function key .
11 Press the vertical soft key [SETTING].
12 Press the MDI switch on the machine operator's panel or enter state emergency stop.
13 Enter 0 in response to the prompt for "PARAMETER WRITE" in setting data.
14 Turn the power of the CNC on again.

Outputting pitch error compensation data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [PITCH] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name "PITCH.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the pitch error compensation data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting pitch error compensation data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [PITCH ERROR] appears.
Press the vertical soft key [PITCH ERROR].
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the file name is omitted, default file name "PITCH.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the pitch error compensation data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3.6 Inputting/Outputting Custom Macro Common Variables

Custom macro common variables can be input and output using the ALL IO screen.

Inputting custom macro common variables (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [MACRO] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "MACRO.TXT" is assumed.

6 Press the soft key [EXEC].
This starts reading the custom macro common variables, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting custom macro common variables (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [MACRO] appears.
Press the vertical soft key [MACRO].
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the horizontal soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name “MACRO.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the custom macro common variables, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Outputting custom macro common variables (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [MACRO] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name “MACRO.TXT” is assumed.
6 Press the soft key [EXEC].
This starts outputting the custom macro common variables, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting custom macro common variables (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [MACRO] appears.
Press the vertical soft key [MACRO].
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the file name is omitted, default file name “MACRO.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the custom macro common variables, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3.7 Inputting and Outputting Workpiece Coordinates System Data

Workpiece coordinates system data can be input and output using the ALL IO screen.

Inputting workpiece coordinate system data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [WORK] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "EXT_WKZ.TXT" is assumed.
6 Press the soft key [EXEC].
This starts reading the workpiece coordinate system data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting workpiece coordinate system data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [WORK] appears.
Press vertical soft key [WORK].
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the horizontal soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name "EXT_WKZ.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the workpiece coordinate system data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Outputting workpiece coordinate system data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [WORK] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name "EXT_WKZ.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the workpiece coordinate system data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting workpiece coordinate system data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [WORK] appears.
Press vertical soft key [WORK].
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the file name is omitted, default file name "EXT_WKZ.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the workpiece coordinate system data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3.8 Inputting and Outputting Operation History Data

Operation history data can be output on the ALL IO screen (with function key 

for 15/19-inch display unit).
Only output operation is permitted for operation history data.
The output data is of text format. So, to reference the output data you must use an application that can handle text files on the personal computer.

Outputting operation history data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [OPERAT HISTRY] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name "OPRT_HIS.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the operation history data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting operation history data (for 15/19-inch display unit)

Procedure

1 Press the function key 

.
2 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [OPERAT HISTRY] appears.
Press the vertical soft key [OPERAT HISTRY].
3 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
4 Press the horizontal soft key [F OUTPUT].
5 Set the file name to be output.
If the file name is omitted, default file name "OPRT_HIS.TXT" is assumed.

6 Press the horizontal soft key [EXEC].
This starts outputting the operation history data, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears. To cancel the output, press the horizontal soft key [CAN].

8.3.9 Inputting and Outputting Tool Management Data

Tool management data can be input and output using the ALL IO screen.

NOTE
1 For multi-path systems, place all paths in the EDIT mode before performing input and output operations.
2 The format used is the same as the registration format of the G10 format.

Inputting tool management data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [TOOL] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name “TOOL_MNG.TXT” is assumed.
6 Press the soft key [EXEC].
This starts reading the tool management data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting tool management data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL] appears.
Press the vertical soft key [TOOL].
2 Press the EDIT switch on the machine operator’s panel.
3 Press the horizontal soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name “TOOL_MNG.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the tool management data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Outputting tool management data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [TOOL] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].

5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the output file name is omitted, default output file name “TOOL_MNG.TXT” is assumed.
6 Press the soft key [EXEC].
This starts outputting the tool management data, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting tool management data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [TOOL] appears.
Press the vertical soft key [TOOL].
2 Press the EDIT switch on the machine operator’s panel.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the output file name is omitted, default output file name “TOOL_MNG.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the tool management data, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Inputting magazine data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [MAGAZINE] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name “MAGAZINE.TXT” is assumed.
6 Press the soft key [EXEC].
This starts reading the magazine data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting magazine data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [MAGAZINE] appears.
Press the vertical soft key [MAGAZINE].
2 Press the EDIT switch on the machine operator’s panel.
3 Press the horizontal soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name “MAGAZINE.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the magazine data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Outputting magazine data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [MAGAZINE] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name "MAGAZINE.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the magazine data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting magazine data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [MAGAZINE] appears.
Press the vertical soft key [MAGAZINE].
2 Press the EDIT switch on the machine operator's panel.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the file name is omitted, default file name "MAGAZINE.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the magazine data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Inputting tool life status name data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [STATUS] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "STATUS.TXT" is assumed.
6 Press the soft key [EXEC].
This starts reading the tool life status name data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting tool life status name data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [STATUS] appears.
Press the vertical soft key [STATUS].
2 Press the EDIT switch on the machine operator's panel.
3 Press the horizontal soft key [N INPUT].

4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name “STATUS.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the tool life status name data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Outputting tool life status name data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [STATUS] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name “STATUS.TXT” is assumed.
6 Press the soft key [EXEC].
This starts outputting the tool life status name data, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the soft key [CAN].

Outputting tool life status name data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [STATUS] appears.
Press the vertical soft key [STATUS].
2 Press the EDIT switch on the machine operator’s panel.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the file name is omitted, default file name “STATUS.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the tool life status name data, and “OUTPUT” blinks in the lower right part of the screen. When the read operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Inputting name data of customize data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [CUSTOM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name “CUSTOMIZ.TXT” is assumed.
6 Press the soft key [EXEC].
This starts reading the name data of customize data, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting name data of customize data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [CUSTOM] appears.
Press the vertical soft key [CUSTOM].
2 Press the EDIT switch on the machine operator's panel.
3 Press the horizontal soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name "CUSTOMIZ.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the name data of customize data, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Outputting name data of customize data (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [CUSTOM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the file name is omitted, default file name "CUSTOMIZ.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the name data of customize data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].

Outputting name data of customize data (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [CUSTOM] appears.
Press the vertical soft key [CUSTOM].
2 Press the EDIT switch on the machine operator's panel.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the file name is omitted, default file name "CUSTOMIZ.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the name data of customize data, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3.10 Inputting and Outputting All Tool Management Data at a Time

All data used by the tool management function is classified into the following two types: “all tool data” and “all custom data” and each type of data is input and output at a time on the ALL IO screen.

Explanation

- Input/output data

Each type of data of the tool management function is input from and output to an external input/output device. The following tables list the types of output data and the data output order.

Table 8.3.10 (a) All tool data

Output order	Output data
1	Tool geometry data
2	Tool management data
3	Magazine management table
4	Magazine property data
5	Pot property data
6	Tool offset data
7	Tool geometry size data

Table 8.3.10 (b) All custom data

Output order	Output data
1	Name data of customize data
2	Tool life status name data
3	Customize data displayed as tool management data
4	Spindle and waiting position name data
5	Decimal point position data of customize data
6	Removal reason data
7	Number of decimal places of magazine property customize data
8	Number of decimal places of pot property customize data
9	Name data of magazine property customize data
10	Name data of pot property customize data
11	Name data of tool type numbers

NOTE

1 For a multi-path system, place all paths in the EDIT mode before inputting and outputting data.
2 After the following tool management data is input normally, turn the power off:
   - Customize data displayed as tool management data
   - Decimal point position data of customize data
   - Number of decimal places of magazine property customize data
   - Number of decimal places of pot property customize data
3 If a target tool is registered in a magazine and interferes with other tools in registration or modification of tool geometry data, alarm PS5360 is issued. In this case, erase the tool geometry data, tool management data, and tool magazine table data in the NC or edit input data, then input all data at a time.
4 If a function related to input tool management data is disabled or the input data format is invalid, input operation is stopped.
5 When creating or editing input data, do not change the order in which data is output at a time.
6 The tool offset data output using this function cannot be used by the programmable data input function.

NOTE
7 For a multi-path system, do not input data output for another path.

Inputting all tool data at a time (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [ALL TOOL] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
6 Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "AL_TLMGR.TXT" is assumed.
7 Press the soft key [EXEC].
This starts reading the data, and "INPUT" blinks in the lower right part of the screen.
When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting all tool data at a time (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [ALL TOOL] appears.
2 Press the vertical soft key [ALL TOOL].
3 Press the EDIT switch on the machine operator's panel.
4 Press the horizontal soft key [N INPUT].
5 Set the name of the file that you want to input.
6 Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "AL_TLMGR.TXT" is assumed.
7 Press the soft key [EXEC].
This starts reading the data, and "INPUT" blinks in the lower right part of the screen.
When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Outputting all tool data at a time (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [ALL TOOL] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
6 Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default output file name "AL_TLMGR.TXT" is assumed.
7 Press the soft key [EXEC].
This starts outputting the data, and "OUTPUT" blinks in the lower right part of the screen.
When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Outputting all tool data at a time (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [ALL TOOL] appears.

2 Press the vertical soft key [ALL TOOL].
3 Press the EDIT switch on the machine operator's panel.
4 Press the horizontal soft key [F OUTPUT].
5 Set the file name to be output.
6 Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default output file name "AL_TLMGR.TXT" is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the data, and "OUTPUT" blinks in the lower right part of the screen.
When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Inputting all custom data at a time (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [ALL CUSTOM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
6 Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "AL_TLCSM.TXT" is assumed.
7 Press the soft key [EXEC].
This starts reading the data, and "INPUT" blinks in the lower right part of the screen.
When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting all custom data at a time (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [ALL CUSTOM] appears.
2 Press the vertical soft key [ALL CUSTOM].
3 Press the EDIT switch on the machine operator's panel.
4 Press the horizontal soft key [N INPUT].
5 Set the name of the file that you want to input.
6 Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name "AL_TLCSM.TXT" is assumed.
7 Press the soft key [EXEC].
This starts reading the data, and "INPUT" blinks in the lower right part of the screen.
When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Outputting all custom data at a time (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [ALL CUSTOM] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
6 Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default output file name "AL_TLCSM.TXT" is assumed.
7 Press the soft key [EXEC].
This starts outputting the data, and "OUTPUT" blinks in the lower right part of the screen.

When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Outputting all custom data at a time (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [ALL CUSTOM] appears.
2 Press the vertical soft key [ALL CUSTOM].
3 Press the EDIT switch on the machine operator’s panel.
4 Press the horizontal soft key [F OUTPUT].
5 Set the file name to be output.
6 Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default output file name “AL_TLCSM.TXT” is assumed.
7 Press the horizontal soft key [EXEC].
This starts outputting the data, and “OUTPUT” blinks in the lower right part of the screen.
When the write operation ends, the “OUTPUT” indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3.11 Inputting and Outputting Workpiece Setting Error Compensation Value

Workpiece setting error compensation value can be input and output using the ALL IO screen.

Inputting workpiece setting error compensation value (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [WORK SET ER] on the ALL IO screen.
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [N INPUT].
5 Set the name of the file that you want to input.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default input file name “WSEC_VAL.TXT” is assumed.
6 Press the soft key [EXEC].
This starts reading the workpiece setting error compensation value, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the soft key [CAN].

Inputting workpiece setting error compensation value (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [WORK SET ER] appears.
Press the vertical soft key [WORK SET ER].
2 Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
3 Press the horizontal soft key [N INPUT].
4 Set the name of the file that you want to input.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default input file name “WSEC_VAL.TXT” is assumed.
5 Press the horizontal soft key [EXEC].
This starts reading the workpiece setting error compensation value, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Outputting workpiece setting error compensation value (for 8.4/10.4-inch display unit)

Procedure

1 Press the soft key [WORK SET ER] on the ALL IO screen.
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the soft key [(OPRT)].
4 Press the soft key [F OUTPUT].
5 Set the file name to be output.
Type a file name, and press the soft key [F NAME].
If the input file name is omitted, default output file name "WSEC_VAL.TXT" is assumed.
6 Press the soft key [EXEC].
This starts outputting the workpiece setting error compensation value, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the soft key [CAN].
7 To cancel the output, press the soft key [CAN].

Outputting workpiece setting error compensation value (for 15/19-inch display unit)

Procedure

1 On the ALL IO screen, press the vertical soft key [NEXT PAGE] until vertical soft key [WORK SET ER] appears.
Press the vertical soft key [WORK SET ER].
2 Press the EDIT switch on the machine operator's panel or enter state emergency stop.
3 Press the horizontal soft key [F OUTPUT].
4 Set the file name to be output.
Type a file name, and press the horizontal soft key [F NAME].
If the input file name is omitted, default output file name "WSEC_VAL.TXT" is assumed.
5 Press the horizontal soft key [EXEC].
This starts outputting the workpiece setting error compensation value, and "OUTPUT" blinks in the lower right part of the screen. When the write operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

8.3.12 File Format and Error Messages

Explanation

- File format

All files that are read from and written to an external device are of text format. The format is described below.
A file starts with % or LF, followed by the actual data. A file always ends with %. In a read operation, data between the first % and the next LF is skipped. Each block ends with an LF, not a semicolon (;).

- LF: 0A (hexadecimal) of ASCII code
- When a file containing lowercase letters, kana characters, and several special characters (such as $, \, and !) is read, those letters and characters are ignored.
Example)

```
%
O0001(MEMORY CARD SAMPLE FILE)
G17 G49 G97
G92 X-11.3 Y2.33
  :
  :
M30
%
```

- ASCII code is used for input/output, regardless of the setting parameter (ISO/EIA).
- Bit 3 (NCR) of parameter No. 0100 can be used to specify whether the end of block code (EOB) is output as "LF" only, or as "LF, CR, CR."

Limitation

- Memory card specification

Use memory cards that comply with PCMCIA Ver2.0 or JEIDA Ver4.1.

- Attribute memory

Memory cards that have no attribute memory or contain no device information in the attribute memory cannot be used.

- Flash ROM card

Flash ROM cards can be used for reading only.

8.4 EMBEDDED ETHERNET OPERATIONS

8.4.1 FTP File Transfer Function

The operation of the FTP file transfer function is described below.

Host file list display (for 8.4/10.4-inch display unit)

A list of the files held on the host computer is displayed.

Procedure

1 Press the function key .
2 Press the soft key [FOLDER]. The program list screen appears. (If the soft key does not appear, press the continuous menu key .)
3 Press the soft key [(OPRT)].
4 Press the soft key [DEVICE CHANGE].
5 Press the soft key [EMB ETHER], and the EMBEDDED ETHERNET HOST FILE LIST screen appears.

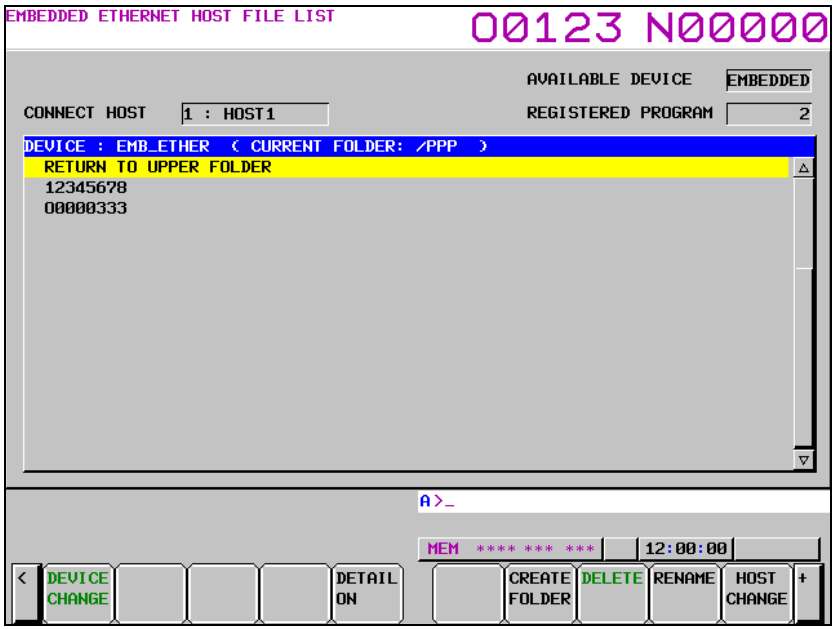


Fig. 8.4.1 (a) Embedded Ethernet host file list screen

NOTE
1 When using the FTP file transfer function, check that the valid device is the embedded Ethernet port.
The two conditions below determine a connection destination on the host file list screen:
(1) Check that the valid device is the embedded Ethernet port. Make a selection in "DEVICE SELECTION" on the Ethernet setting screen.
(2) A host computer can be selected from connection destinations 1, 2, and 3. Make a selection according to soft key [HOST CHANGE] described later.
2 File names including 2-bytes characters cannot be displayed correctly.

6 When a list of files is larger than one page, the screen display can be switched using the page keys PAGE PAGE.
7 Press the soft key [REFRESH] to update the screen display.
8 Press the soft key [DETAIL OFF] to display the host file list (file names only) screen. Press the soft key [DETAIL ON] to display the host file list (detail) screen.
9 When selecting a folder with cursor and pressing the MDI key INPUT, the selected folder is changed to the new current folder.

Host file list display (for 15/19-inch display unit)

A list of the files held on the host computer is displayed.

Procedure

1 Press the function key PROG.
2 Press the vertical soft key [FOLDER]. The program list screen appears.
3 Press the continuous menu key ▷ until horizontal soft key [DEVICE CHANGE] appears. Press the horizontal soft key [DEVICE CHANGE].
4 Press the horizontal soft key [EMB ETHER], and the EMBEDDED ETHERNET HOST FILE LIST screen appears.

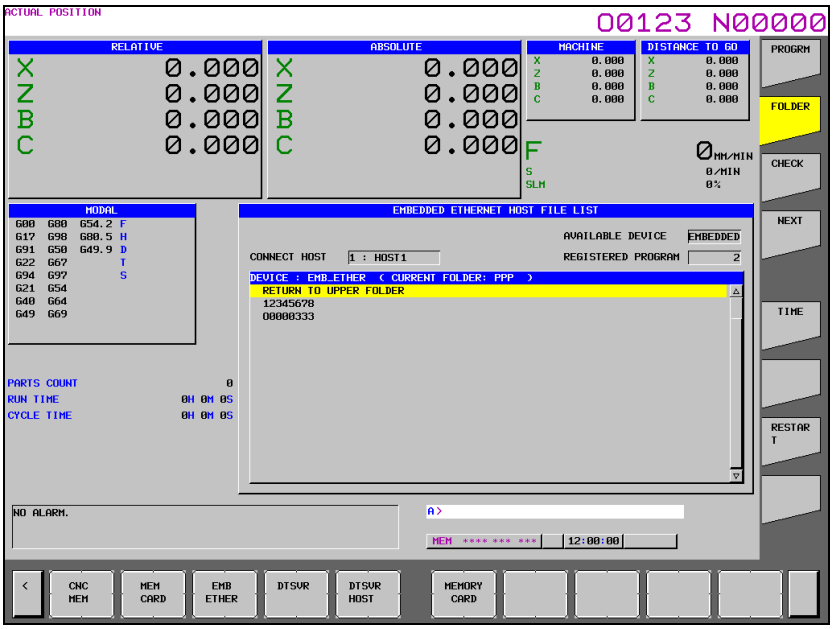


Fig. 8.4.1 (b) Embedded Ethernet host file list screen

NOTE

1 When using the FTP file transfer function, check that the valid device is the embedded Ethernet port.
The two conditions below determine a connection destination on the host file list screen:
(1) Check that the valid device is the embedded Ethernet port. Make a selection in "DEVICE SELECTION" on the Ethernet setting screen.
(2) A host computer can be selected from connection destinations 1, 2, and 3. Make a selection according to soft key [HOST CHANGE] described later.

2 File names including 2-bytes characters cannot be displayed correctly.

5 When a list of files is larger than one page, the screen display can be switched using the page keys PAGE PAGE.

6 Press the horizontal soft key [REFRESH] to update the screen display.

7 Press the horizontal soft key [DETAIL OFF] to display the host file list (file names only) screen. Press the horizontal soft key [DETAIL ON] to display the host file list (detail) screen.

8 When selecting a folder with cursor and pressing the MDI key INPUT, the selected folder is changed to the new current folder.

Display item (common to 8.4/10.4/15/19-inch display units)

AVAILABLE DEVICE

"EMBEDDED" or "PCMCIA" is displayed.

CONNECT HOST

The currently selected connection host number and host name are displayed.

REGISTERED PROGRAM

The number of files registered in the work folder of the connected host is displayed. Up to 8 digits can be displayed.

CURRENT FOLDER

The current folder name of the connected host is displayed.
If the folder-path is long compared with the display-item, characters: "…" and only the first and last ten letters of the folder name are displayed.

FILE LIST

There is no distinction between file names and folder names.
Although the maximum number of displayed characters is 127, characters are displayed as far as they can be displayed in one line.

Operation list (common to 8.4/10.4/15/19-inch display units)

DETAIL ON, DETAIL OFF

The screen display can be switched between the display of file names only and the display of details.

REFRESH

Display data can be updated.

F INPUT

A file can be input from the host computer to the part program storage memory of the CNC. This item is displayed only when 9 is set as the I/O device number of the CNC.

F OUTPUT

A file can be output from the part program storage memory of the CNC to the host computer. This item is displayed only when 9 is set as the I/O device number of the CNC.

DELETE

A folder or file can be deleted from the host computer.

RENAME

A file or folder on the host computer can be renamed.

CREATE FOLDER

A folder can be created on the host computer.

HOST CHANGE

The connected host can be changed.

NC program input (for 8.4/10.4-inch display unit)

A file (NC program) stored on the host computer can be input into the part program storage memory of the CNC.

Procedure

1 Display the EMBEDDED ETHERNET HOST FILE LIST screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the continuous menu key ▷ until soft key [F INPUT] appears.
Press the soft key [F INPUT].
5 Position the cursor on the file to input, and press [F GET] or type the file name to input.
Press the soft key [F SET].
If the input file name is omitted, default input file name "ALL-PROG.TXT" is assumed.

6 Type the program name, and press the soft key [P SET].
For an explanation of the operations to be performed if an input file name [F SET] and an input program name [P SET] are omitted, see the Table 8.4.1 (a).
7 Press the soft key [EXEC].
This starts reading the program, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the soft key [CAN].

Table 8.4.1 (a)

[F SET]	[P SET]	Input file name	Input program	Input program name
BLANK	BLANK	ALL-PROG.TXT	All programs in ALL-PROG.TXT	File name at the time the file is saved
BLANK	INPUT	ALL-PROG.TXT	First program in ALL-PROG.TXT	Program name set with [P SET]
INPUT	BLANK	File name set with [F SET]	All programs in the file specified with [F SET]	File name at the time the file is saved
INPUT	INPUT	File name set with [F SET]	First program in the file specified with [F SET]	Program name set with [P SET]

NC program input (for 15/19-inch display unit)

A file (NC program) stored on the host computer can be input into the part program storage memory of the CNC.

Procedure

1 Display the EMBEDDED ETHERNET HOST FILE LIST screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the continuous menu key  until horizontal soft key [F INPUT] appears.
Press the horizontal soft key [F INPUT].
4 Position the cursor on the file to input, and press [F GET] or type the file name to input.
Press the horizontal soft key [F SET].
If the input file name is omitted, default input file name "ALL-PROG.TXT" is assumed.
5 Type the program name, and press the horizontal soft key [P SET].
For an explanation of the operations to be performed if an input file name [F SET] and an input program name [P SET] are omitted, see the Table 8.4.1 (b).
6 Press the horizontal soft key [EXEC].
This starts reading the program, and "INPUT" blinks in the lower right part of the screen. When the read operation ends, the "INPUT" indication disappears.
To cancel the input of the program, press the horizontal soft key [CAN].

Table 8.4.1 (b)

[F SET]	[P SET]	Input file name	Input program	Input program name
BLANK	BLANK	ALL-PROG.TXT	All programs in ALL-PROG.TXT	File name at the time the file is saved
BLANK	INPUT	ALL-PROG.TXT	First program in ALL-PROG.TXT	Program name set with [P SET]
INPUT	BLANK	File name set with [F SET]	All programs in the file specified with [F SET]	File name at the time the file is saved
INPUT	INPUT	File name set with [F SET]	First program in the file specified with [F SET]	Program name set with [P SET]

NC program output (for 8.4/10.4-inch display unit)

A file (NC program) stored in the part program storage memory of the CNC can be output to the host computer.

Procedure

1 Display the EMBEDDED ETHERNET HOST FILE LIST screen.

2 Press the EDIT switch on the machine operator's panel.
3 Press the soft key [(OPRT)].
4 Press the continuous menu key ▷ until soft key [F OUTPUT] appears.
Press the soft key [F OUTPUT].
5 Type the program name to output, and press the soft key [P SET]. To specify an output file name, type the output file name and press the soft key [F SET].
If, in this step, no output file name or no program name is specified, all programs in the foreground folders are output to "ALL-PROG.TXT".
For an explanation of the operations to be performed if an output file name [F SET] and an output program name [P SET] are omitted, see the Table 8.4.1 (c).
(For an explanation of foreground folders, see II-12.1.3, and for how to change foreground folders, see III-11.6)
6 Press the soft key [EXEC].
This starts outputting the program, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output of the program, press the soft key [CAN].

Table 8.4.1 (c)

[F SET]	[P SET]	Output file name	Output program
BLANK	BLANK or (O-9999)	ALL-PROG.TXT	All programs in the foreground folders displayed in the program folder
BLANK	INPUT	Program name set with [P SET]	Program in the NC that is set with [P SET]
INPUT	BLANK or (O-9999)	File name set with [F SET]	All programs in the foreground folders displayed in the program folder
INPUT	INPUT	File name set with [F SET]	Program in the NC that is set with [P SET]

NC program output (for 15/19-inch display unit)

A file (NC program) stored in the part program storage memory of the CNC can be output to the host computer.

Procedure

1 Display the EMBEDDED ETHERNET HOST FILE LIST screen.
2 Press the EDIT switch on the machine operator's panel.
3 Press the continuous menu key ▷ until horizontal soft key [F OUTPUT] appears.
Press the horizontal soft key [F OUTPUT].
4 Type the program name to output, and press the horizontal soft key [P SET]. To specify an output file name, type the output file name and press the horizontal soft key [F SET].
If, in this step, no output file name or no program name is specified, all programs in the foreground folders are output to "ALL-PROG.TXT".
For an explanation of the operations to be performed if an output file name [F SET] and an output program name [P SET] are omitted, see the Table 8.4.1 (d).
(For an explanation of foreground folders, see II-12.1.3, and for how to change foreground folders, see III-11.6)
5 Press the horizontal soft key [EXEC].
This starts outputting the program, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
To cancel the output, press the horizontal soft key [CAN].

Table 8.4.1 (d)

[F SET]	[P SET]	Output file name	Output program
BLANK	BLANK or (O-9999)	ALL-PROG.TXT	All programs in the foreground folders displayed in the program folder
BLANK	INPUT	Program name set with [P SET]	Program in the NC that is set with [P SET]

[F SET]	[P SET]	Output file name	Output program
INPUT	BLANK or (O-9999)	File name set with [F SET]	All programs in the foreground folders displayed in the program folder
INPUT	INPUT	File name set with [F SET]	Program in the NC that is set with [P SET]

8.5 SCREEN HARD COPY FUNCTION

Overview

This function converts screen information displayed on the CNC into bit map format data and output it to a memory card or USB memory. Once output, bit map format data can be displayed and edited on a personal computer. This function can be enabled by setting bit 7 (HDC) of parameter No. 3301.

Explanation

- Start/cancellation methods

The screen hard copy function is started by pressing and holding down key 

for five seconds. The function can be canceled by pressing key 

.

- Acquisition and output of screen data

When started, the screen hard copy function starts acquiring screen data. As soon as it has acquired it, the function outputs bit map format data to the memory card inserted into the LCD unit. While screen data is being acquired, the screen remains stationary for a few seconds. Acquired screen data can be output from the memory card screen. Also, while screen data is being output, "OUTPUT" blinks in the status display.

- Screen data file names

Bit map format screen data files created by this function are assigned the names below, starting with the one created after the power is turned on.
"HDCPY000.BMP" (name of the first file output to the memory card after the power is turned on)
"HDCPY001.BMP" (name of the second file output to the memory card after the power is turned on)
:
"HDCPY999.BMP" (name of the 1000th file output to the memory card after the power is turned on)
If, after a file with "HDCPY999.BMP" is output, the screen hard copy function is executed, the file name returns to "HDCPY000.BMP". If, however, a file with the same file name as the one to be output when the screen hard copy function is executed exists on the memory card, alarm SR1973 is issued. If the capacity of the memory card is exceeded, alarm SR1962 is issued.
In either case, screen data is not output, so that either the existing file needs to be deleted or the memory card needs to be replaced with a new one.

When a USB memory is used, if the USB memory contains a file with the same file name as the one to be output by executing the screen hard copy function, the file is overwritten. If the capacity of the USB memory is exceeded, alarm SR1932 is issued. In this case, the existing file needs to be deleted or the USB memory needs to be replaced with a new one.

Limitation

- Screens whose hard copies cannot be made

Hard copies of the BOOT screen, the IPL screen, and the system alarm screen cannot be made.

- Foreground I/O devices

During DNC operation, for example, screen data cannot be output while a foreground I/O device is in use.

- Canceling the hard copy function

If the hard copy function is canceled before a hard copy is completed, an incomplete bit map file of data that has been output is created.

- CNC screen display function

If CNC screen display function is working, the screen hard copy function is disabled. In this case, bit map format data can be made by screen copy function of CNC screen display function. Please refer to the operator's manual of CNC screen display function.

- Output folder

The screen data is output to the root folder of the memory card or the USB memory.

8.6 USB FUNCTION

Overview

Setting NC parameter No. 20 to 17 selects a USB memory as the data input/output destination.
Basic specifications of the USB function are listed below.

- The USB function supports the FAT12, FAT16, FAT32, and VFAT file systems.
- It can handle up to six folder levels under the root folder.
- Up to 32 ASCII code characters (alphanumeric characters) are available for a file or folder name.

⚠ CAUTION
A commercially available USB memory does not have sufficient reliability in the FA environment. When a part program, a parameter, etc. are input or output to a USB memory, those data might be damaged and there is a possibility that a serious accident or loss occurs.
Therefore, set bit 0 (ISU) of parameter No.11505 to input or output data to a USB memory using ISO codes unless data is input using ASCII codes.

NOTE
1 Do not turn off CNC or do not remove the USB memory during accessing the USB memory. The USB memory may be damaged.
2 There is a USB memory not designed for the long preservation of data. Therefore, do not use a USB memory by the long preservation purpose of data, and back up important data to the personal computer.
3 When empty capacity of the USB memory decreases or the number of files in the USB memory increases, it might take time to the input or output operation. In this case, input or output after deleting an unnecessary file in the USB memory.
4 Any USB memory that is not physically formatted cannot be formatted on the CNC. Format a USB memory to be used for the first time with FAT or FAT32 without specifying the Quick Format option on the personal computer.

Restrictions

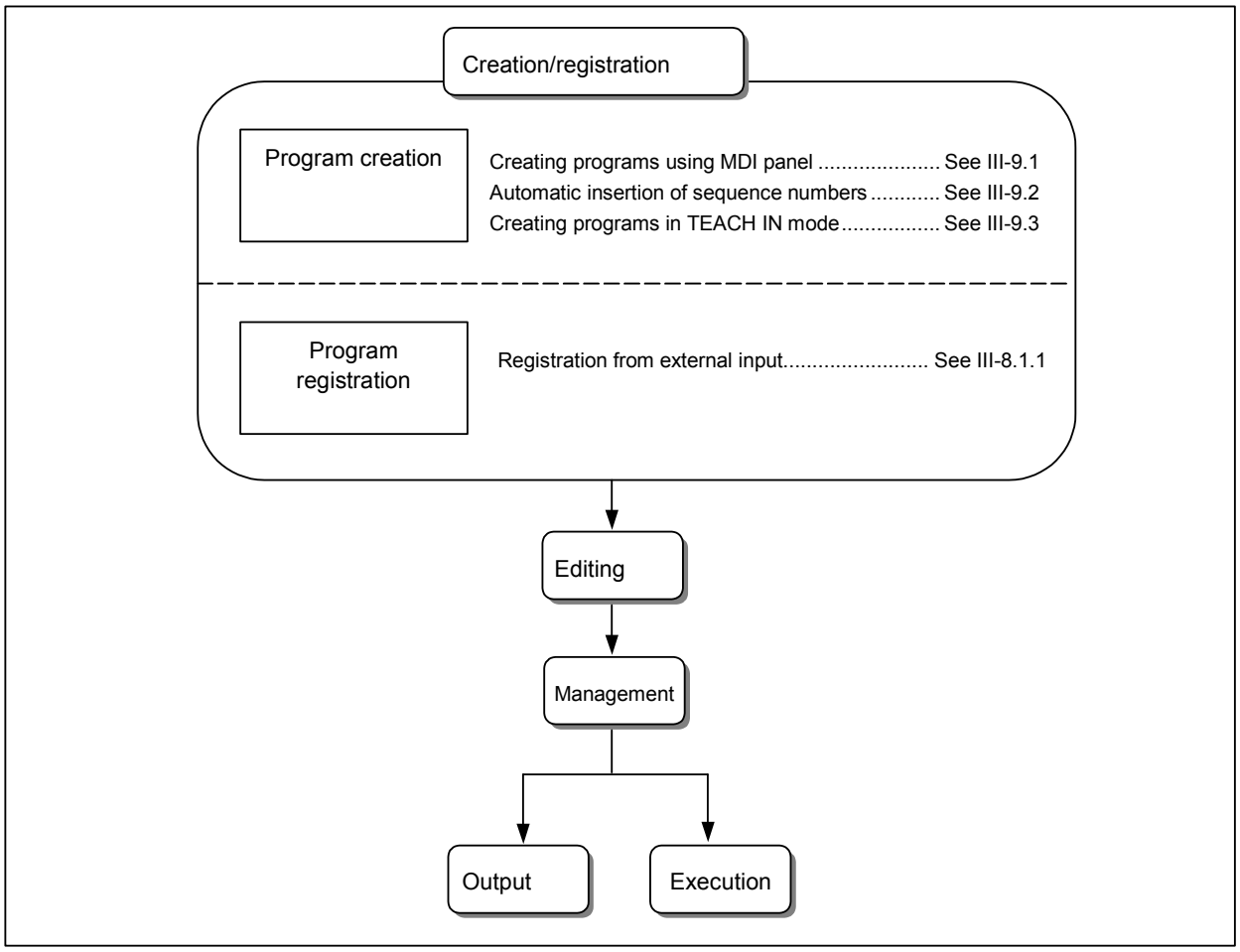
There are the following restrictions on the USB function.

- Use a USB memory conforming to USB Ver. 1.0. All of USB memory operation is not guaranteed, however. Before using a USB memory, fully check its operation.
- Only one USB memory can be recognized. (No USB hub can be recognized.)

- DNC operation cannot be performed using a program in a USB memory.
- Schedule operation cannot be performed using a program in a USB memory.
- A program in a USB memory device cannot be called using an external subprogram call (M198).
- "Memory Card Program Operation/Editing" cannot be applied to a USB memory.

9 CREATING PROGRAMS

This chapter explains how to create programs by MDI of the CNC.
This chapter also explains automatic insertion of sequence numbers and how to create programs in TEACH IN mode.



9.1 CREATING PROGRAMS USING THE MDI PANEL

Programs can be created in the EDIT mode using the program editing functions described in III-10.

Procedure for Creating Programs Using the MDI Panel

1 Enter the EDIT mode.
2 Press the PROG key.
3 Press address key P 0 and enter the program number.
4 Press the INSERT key.
5 Create a program using the program editing functions described in III-10.

Explanation

- Comments in a program

Comments can be written in a program using the control in/out codes.
Example) O0001 (TEST PROGRAM) ;
M08 (COOLANT ON) ;

- When the INSERT key is pressed after the control-out code "(", comments, and control-in code ")" have been typed, the typed comments are registered.
- When the INSERT key is pressed midway through comments, to enter the rest of comments later, the data typed before the INSERT key is pressed may not be correctly registered (not entered, modified, or lost) because the data is subject to an entry check which is performed in normal editing.
- Control-out code "(" or Control-in code ")" cannot be registered by itself.

9.2 AUTOMATIC INSERTION OF SEQUENCE NUMBERS

Sequence numbers can be automatically inserted in each block when a program is created using the MDI keys in the EDIT mode.
Set the increment for sequence numbers in parameter No. 3216.

Procedure for automatic insertion of sequence numbers

Procedure

1 Enter 1 in response to the prompt for "SEQUENCE NO." in setting data. (see III-12.3.1).
2 Enter the EDIT mode.
3 Press PROG key to display the program screen.
4 Search for or register the number of a program to be edited and move the cursor to the EOB (;) of the block after which automatic insertion of sequence numbers is started.
When a program number is registered and an EOB (;) is entered with the INSERT key, sequence numbers are automatically inserted starting with 0. Change the initial value, if required, according to step 10, then skip to step 7.
5 Press address key O N and enter the initial value of N.
6 Press INSERT key.
7 Enter each word of a block.
8 Press E EOB key.
9 Press INSERT key. The EOB is registered in memory and sequence numbers are automatically inserted. For example, if the initial value of N is 10 and the parameter for the increment is set to 2, N12 inserted and displayed below the line where a new block is specified.

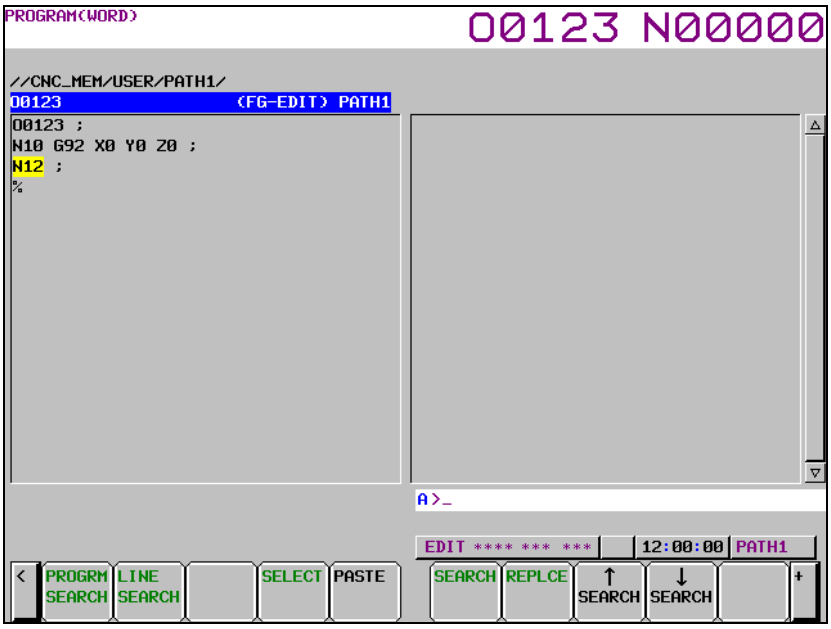


Fig. 9.2 (a)

10 In the example above, if N12 is not necessary in the next block, pressing the DELETE key after N12 is displayed deletes N12.
If wishing to insert N100, not N12, into the next block, type N100 immediately after N12 is displayed, and press CALC ALTER. This causes N100 to be registered and the initial value to be changed to 100.

9.3 CREATING PROGRAMS IN TEACH IN MODE (PLAYBACK)

In the TEACH IN JOG or TEACH IN HANDLE mode, you can create a program while inserting the coordinate of the current position along each axis in the absolute coordinate system when the tool is moved by manual operation into the program.
You can input the words other than axis names in the same way as in the EDIT mode.

Program screen in the TEACH IN JOG mode

Displayed items

In the TEACH IN JOG or TEACH IN HANDLE MODE, the following program screen is displayed.
On the left of the screen, the coordinates of the current position in the absolute and relative coordinate systems are displayed; on the right of the screen, the contents of a program are displayed. You can create a program while checking the current position by manual operation.

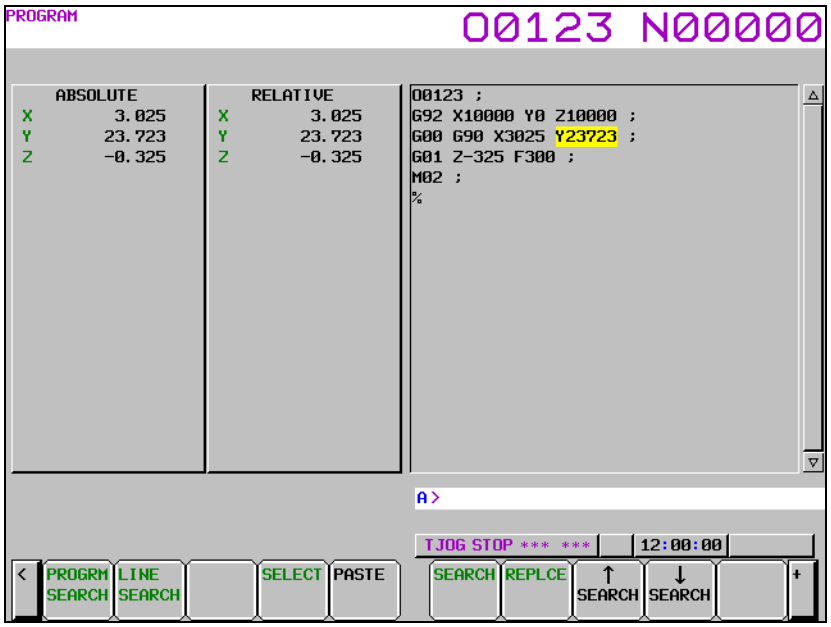


Fig. 9.3 (a) Program screen in the TEACH IN JOG mode

Inputting the coordinates of the current position

You can use the following procedure to insert the coordinate of the current position along each axis in the absolute coordinate system:

1 Select the TEACH IN JOG mode or TEACH IN HANDLE mode.

2 Press PROG key to display the program screen. Search for or register the number of a program to be edited and move the cursor to the position where the current position along each axis is to be inserted.

3 Move the tool to the desired position with jog or handle.

4 Key in the axis name of an axis along which you want to insert the coordinate of the current position.

5 Press the INSERT key. Then, the coordinate of the current position along the specified axis is inserted in the program.
(Example) X10.521 Coordinate of the current position
X10521 Data inserted in the program

Example

```
O1234 ;
G92 X10000 Y0 Z10000 ;
G00 G90 X3025 Y23723 ;
G01 Z-325 F300 ;
M02 ;
```

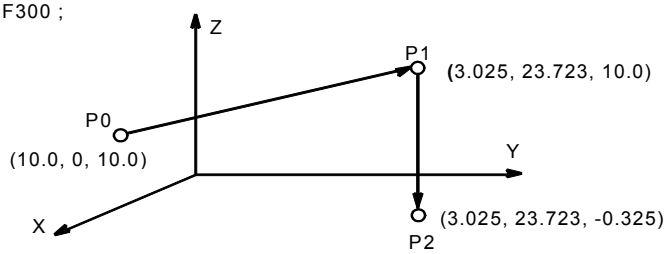
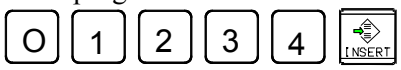


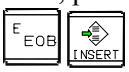
Fig. 9.3 (b)

1 Select the TEACH IN HANDLE mode.
2 Make positioning at position P0 by the manual pulse generator.
3 Select the program screen.

4 Enter program number O1234 as follows:



This operation input program number O1234 in memory.
Next, press the following keys:



An EOB (;) is entered after program number O1234.

5 Enter the P0 machine position for data of the first block as follows:



This operation registers G92X10000Y0Z10000; in program.

6 Position the tool at P1 with the manual pulse generator.

7 Enter the P1 machine position for data of the second block as follows:



This operation input G00G90X3025Z23723; in program.

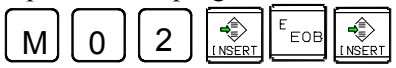
8 Position the tool at P2 with the manual pulse generator.

9 Enter the P2 machine position for data of the third block as follows:



This operation input G01Z -325F300; in program.

10 Input M02; in program as follows:



This completes the registration of the sample program.

Explanation

- Registering a position with compensation

When an axis name and a numeric value are keyed in and the 

key is pressed, the value keyed in is added to the absolute coordinate of the current position and the coordinate is inserted. This operation allows insertion of a corrected value for the absolute coordinate position.
When an extended axis name is used and the name ends with a numeric value, insert an equal sign (=) between the axis name and numeric value.

- Registering commands other than position commands

Commands to be entered before and after a position command must be entered before and after the machine position is registered, by using the same operation as program editing in EDIT mode.

- Calculator-type input

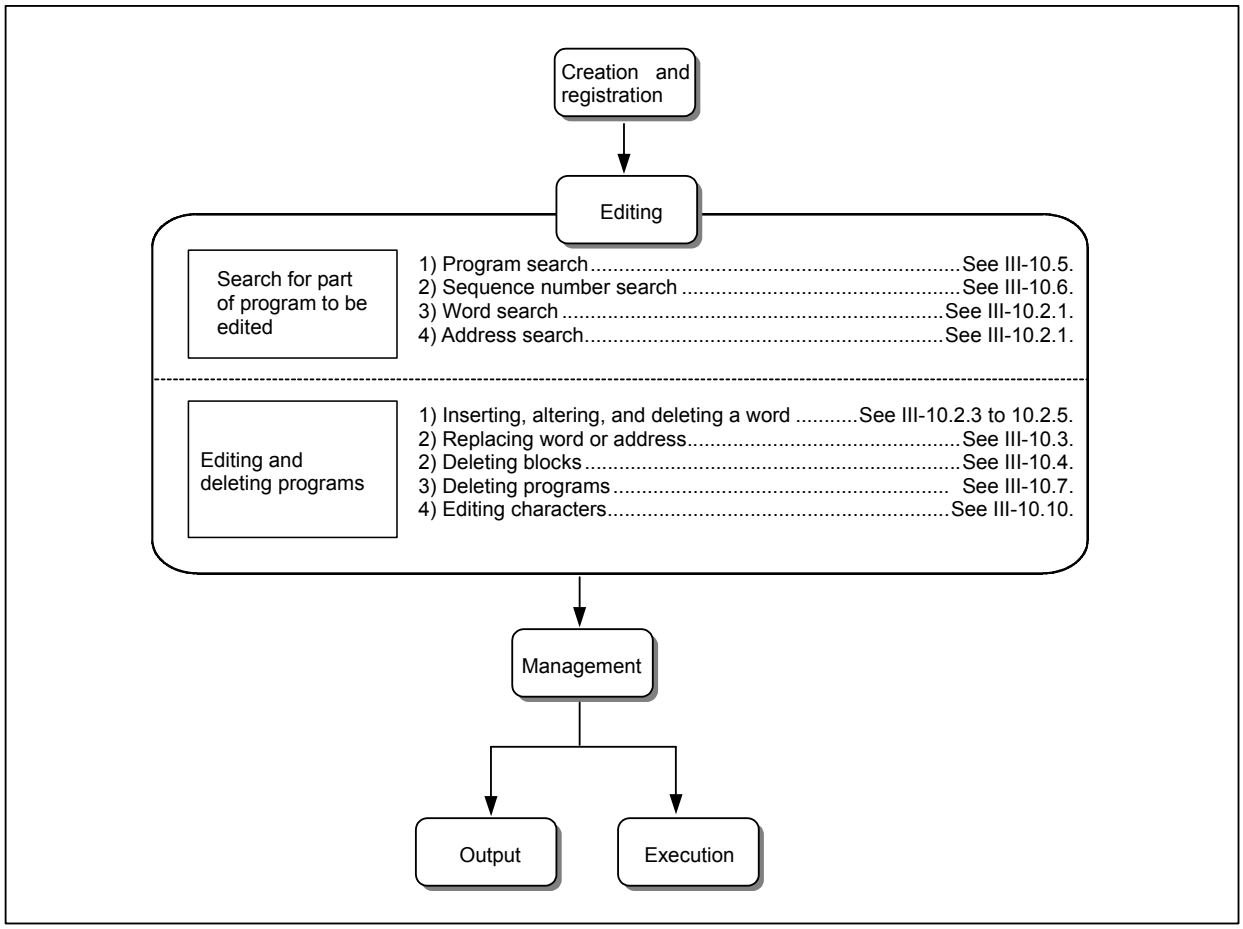
When the calculator-type input format is disabled (bit 0 (DPI) of parameter No. 3401 is set to 0), the coordinate of the current position is inserted into the program in least input increments. When the calculator-type input format is enabled (the bit is set to 1), the coordinate is inserted with a decimal point.

(Example)

Coordinate of the current position X10.521
At this time, the X-axis coordinate is inserted into the program as follows:
When the calculator-type input format is disabled X10521
When the calculator-type input format is enabled X10.521

10 EDITING PROGRAMS

This chapter describes how to edit programs registered in the CNC. Editing includes the insertion, modification, and deletion of words. Editing also includes deletion of the entire program and automatic insertion of sequence numbers. In addition, PC-like program text copy and move operations are possible. This chapter also describes program search, sequence number search, word search, and address search, which are performed before editing the program.



10.1 EDIT DISABLE ATTRIBUTE AND DISPLAY DISABLE ATTRIBUTE

Before a program can be edited, the edit disable attribute and display disable attribute must be removed. The edit disable attribute and display disable attribute can be set for each program and folder.
Programs with the edit disable attribute or display disable attribute, and programs in folders with the edit disable attribute or display disable attribute cannot be edited.

Procedure for removing the edit disable attribute

1 Select EDIT mode.
2 Press the function key 

.
3 Press the soft key [FOLDER].
4 Press the soft key [(OPRT)].
5 Press the soft key [DETAIL ON].

6 Move the cursor to the program or folder from which you want to remove the edit disable attribute.
7 Press the soft key [CHANGE ATTR].
8 Press the soft key [EDIT ENABLE].
9 Press the soft key [END].

⚠ CAUTION
1 After completing editing, set the edit disable attribute as necessary.
2 To set the edit disable attribute, follow the same procedure as for removing the attribute.
In the step 8, press the soft key [EDIT DISABL].

Procedure for removing the display disable attribute

1 Select MDI mode.
2 Set the same value as the password (parameter No. 3210 (PSW)) in parameter No. 3211 (KEY) to release protection of programs.
3 Select EDIT mode.
4 Press the function key 

.
5 Press the soft key [FOLDER].
6 Press the soft key [(OPRT)].
7 Press the soft key [DETAIL ON].
8 Move the cursor to the program or folder from which you want to remove the display disable attribute.
9 Press the soft key [CHANGE ATTR].
10 Press the soft key [DISP ENABLE].
11 Press the soft key [END].

⚠ CAUTION
1 After completing editing, set the display disable attribute as necessary.
2 To set the display disable attribute, perform steps 1 to 9 of the procedure for removing the attribute and press soft key [DISP DISABL] in step 10. Then, set parameter No. 3210 (PSW) to a value other than 0 that is not the value set in parameter No. 3211 (KEY). The program or folder is not displayed in any list screen.

10.2 INSERTING, ALTERING AND DELETING A WORD

This section outlines the procedure for inserting, altering, and deleting a word in a program registered in memory.

Procedure for inserting, altering and deleting a word

1 Select EDIT mode.
2 Press the function key 

.
3 Select a program to be edited.
If a program to be edited is selected, perform the operation 4.
If a program to be edited is not selected, search for the program number.
4 Move the cursor to the position to be edited, with one of the methods below.
- Scan method
- Word search method
- Address search method

For each method, refer to III-10.2.1.

5 Perform an operation such as altering, inserting, or deleting a word.

Explanation

- Concept of word and editing unit

A word is an address followed by a number. With a custom macro, the concept of word is ambiguous. So the editing unit is considered here.

The editing unit is a unit subject to alteration or deletion in one operation. In one scan operation, the cursor indicates the start of an editing unit.

An insertion is made after an editing unit.

Definition of editing unit

- Program portion from an address to immediately before the next address
- An address is an alphabet, IF, WHILE, GOTO, END, DO=, or ; (EOB).

According to this definition, a word is an editing unit.

The word "word," when used in the description of editing, means an editing unit according to the precise definition.

> ⚠ **WARNING**
> When a change, insertion, or deletion was performed on data of a program by pausing machining with the single block stop, feed hold, or other operations during execution of a program, be sure to return the cursor to the original position before restarting the program. To execute the program with the cursor positioned at another position, be sure to make a reset.
> Otherwise, the program may not be executed as expected from the program shown on the screen after machining restarts.

10.2.1 Word Search

A word can be searched for by merely moving the cursor through the text (scanning), by word search, or by address search.

Procedure for scanning a program

1 Press the cursor key [→].

The cursor moves forward word by word; the cursor is displayed at a selected word.

2 Press the cursor key [←].

The cursor moves backward word by word; the cursor is displayed at a selected word.

Example: Scanning Z1250.0

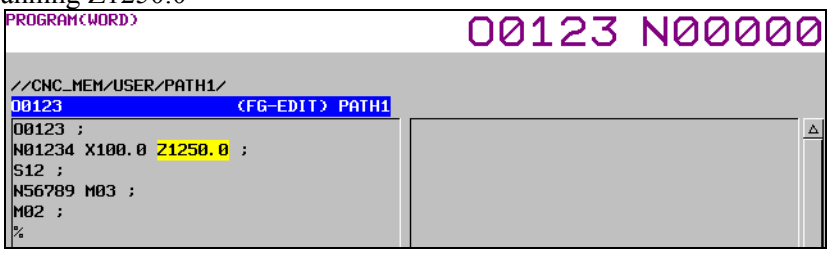


Fig. 10.2.1 (a)

3 Holding down the cursor key [→] or [←] scans words continuously.

4 The first word of the next block is searched for when the cursor key [↓] is pressed.

5 The first word of the previous block is searched for when the cursor key [↑] is pressed.

6 Holding down the cursor key [↓] or [↑] moves the cursor to the head of a block continuously.

7 Pressing the page key [PAGE ↓] displays the next page and searches for the first word of the page.

8 Pressing the page key [↑ PAGE] displays the previous page and searches for the first word of the page.

9 Holding down the page key [PAGE ↓] or [↑ PAGE] displays one page after another.

Procedure for searching a word

Example of searching for S12

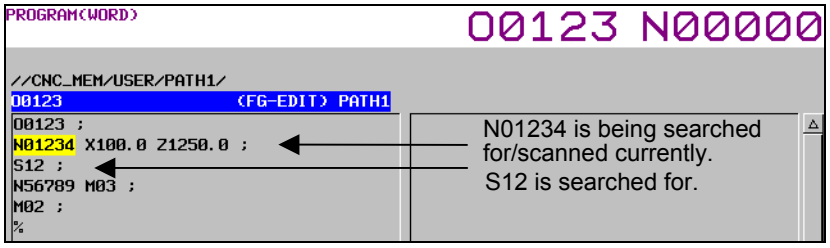


Fig. 10.2.1 (b)

1 Press soft key [SEARCH].

2 Key in address [S].

3 Key in [1] [2].

- S12 cannot be searched for if only S1 is keyed in.
- S09 cannot be searched for by keying in only S9.
  To search for S09, be sure to key in S09.

4 Pressing soft key [↓ SEARCH] starts searching forward from the cursor position. Pressing [↑ SEARCH] starts searching backward.

5 To search for the same word successively, press [↓ SEARCH] or [↑ SEARCH].

Procedure for searching an address

Example of searching for M03

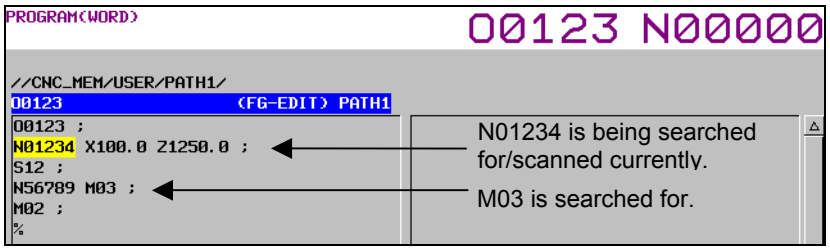


Fig. 10.2.1 (c)

1 Pressing the [SEARCH] key

2 Key in address [M].

3 Pressing soft key [↓ SEARCH] starts searching forward from the cursor position. Pressing [↑ SEARCH] starts searching backward.

4 To search for the same word successively, press [↓ SEARCH] or [↑ SEARCH].

10.2.2 Heading a Program

The cursor can be jumped to the top of a program. This function is called heading the program pointer. This section describes the four methods for heading the program pointer.

Procedure for heading a program

Method 1

1 Press RESET key when the program screen is selected in EDIT mode. When the cursor has returned to the start of the program, the contents of the program are displayed from its start on the screen.

Method 2 Search for the program number.

1 When the program screen is selected in MEMORY or EDIT mode, enter a program number or program name.

When entering a program number, press address key O then type the program number.

2 Press soft key [PROGRM SEARCH].

Method 3

1 Select the program screen or program check screen in MEMORY mode.
2 Press soft key [(OPRT)].
3 Press soft key [REWIND].

Method 4

1 Select the program screen in EDIT mode.
2 Press soft key [(OPRT)].
3 Press soft key [LINE SEARCH].
4 Press soft key [TOP].

10.2.3 Inserting a Word

Procedure for inserting a word

1 Search for or scan the word immediately before a word to be inserted.
2 Key in an address to be inserted.
3 Key in data.
4 Press the INSERT key.

Example of Inserting T15

1 Search for or scan Z1250.

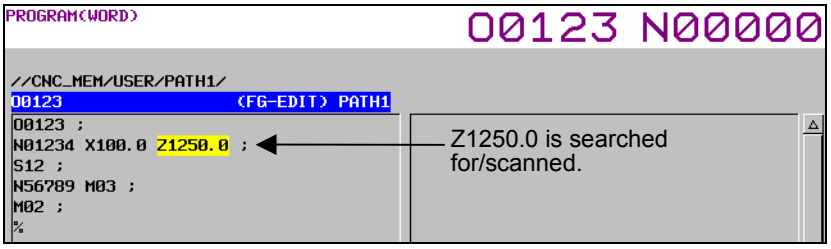


Fig. 10.2.3 (a)

2 Key in T 1 5 .

3 Press the INSERT key.

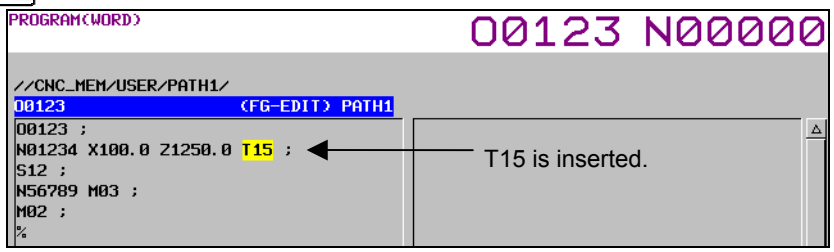


Fig. 10.2.3 (b)

10.2.4 Altering a Word

Procedure for altering a word

1 Search for or scan a word to be altered.
2 Key in an address to be inserted.
3 Key in data.
4 Press the CALC ALTER key.

Example of changing T15 to M15

1 Search for or scan T15.

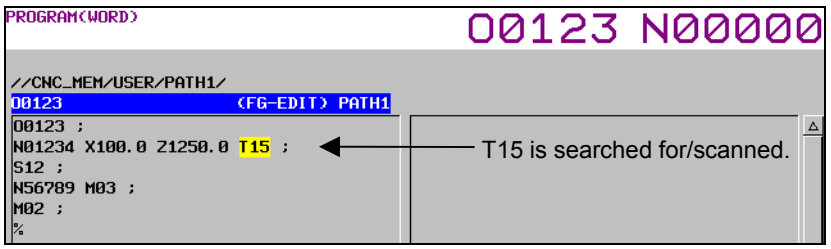


Fig. 10.2.4 (a)

2 Key in M 1 5 .
3 Press the CALC ALTER key.

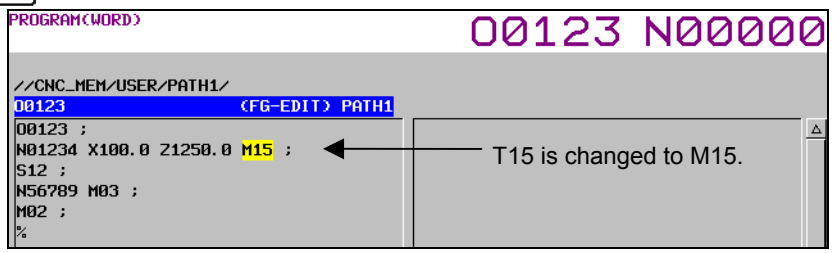


Fig. 10.2.4 (b)

10.2.5 Deleting a Word

Procedure for deleting a word

1 Search for or scan a word to be deleted.
2 Press the DELETE key.

Example of deleting X100.0

1 Search for or scan X100.0.

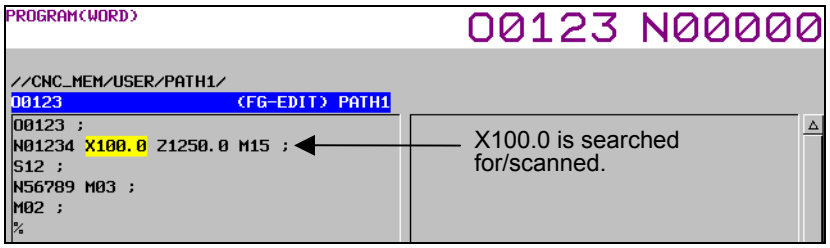


Fig. 10.2.5 (a)

2 Press the DELETE key.

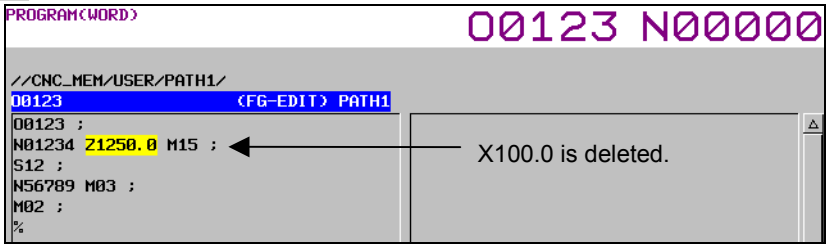


Fig. 10.2.5 (b)

10.3 REPLACING A WORD OR ADDRESS

Replace the specified word or address in a program at a time.
The following methods are available: Replacement after checking each found character string and batch replacement without checking any found character string.

Procedure for replacing a word or address

- Replacing a word or address after checking each found character string

1 Move the cursor to the position from which to start replacement.
2 Press soft key [REPLCE].
3 Enter a word or address before replacement.
4 Press soft key [BEFORE].
5 Enter a word or address after replacement.
6 Press soft key [AFTER].
7 Press soft key [↑ SEARCH] or [↓ SEARCH]. The word or address is searched for in the direction of the arrow.
8 Press soft key [REPLCE] to replace the character string at the cursor and search for the match.
If you want to search for the next match without replacing the character string at the cursor, press soft key [↑ SEARCH] or [↓ SEARCH].
9 Repeat step 8 until required replacement is terminated.
To terminate replacement during operation, press soft key [END].

- Replacing a word or address without checking any found character string

1 Move the cursor to the position from which to start replacement.
2 Press soft key [REPLCE].
3 Enter a word or address before replacement.
4 Press soft key [BEFORE].
5 Enter a word or address after replacement.
6 Press soft key [AFTER].

7 Press soft key [↑ SEARCH] or [↓ SEARCH]. The word or address is searched for in the direction of the arrow.
8 Press soft key [REPLCE ALL].
9 To cancel replace all operation, press soft key [NO].
To execute replace all operation, press soft key [YES]. The word or address is searched for from the cursor position in the direction of the arrow in the soft key pressed in step 7 and all found character strings are replaced at a time. Replace all operation is performed to the beginning of the program for upward replacement or to the end of the program for downward replacement.

NOTE
1 Any character string containing EOB cannot be specified as the character string before or after replacement.
2 Pressing soft key [REPLCE ALL] during replacement with checking each found character string allows replacement without checking the following found character strings to start.

10.4 DELETING BLOCKS

A block or blocks can be deleted in a program.

10.4.1 Deleting a Block

The portion from the current word position to the next EOB is deleted. The cursor is then placed in the word next to the deleted EOB.

Procedure for deleting a block

1 Search for or scan address N for a block to be deleted.
2 Press the [EOB] key.
3 Press the editing key [DELETE].

Example of deleting a block of N01234

1 Search for or scan N01234.

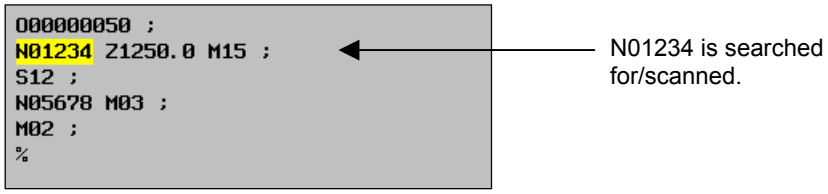


Fig. 10.4.1 (a)

2 Press the [EOB] key.
3 Press the editing key [DELETE].

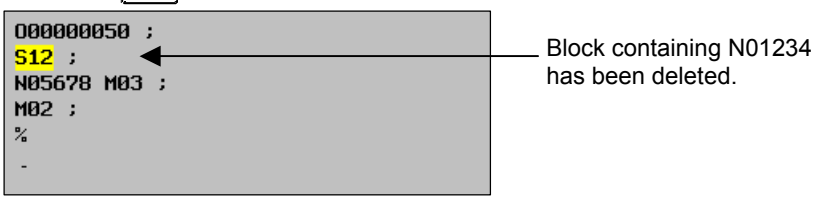


Fig. 10.4.1 (b)

10.4.2 Deleting Multiple Blocks

The several blocks in the forward direction from the current word position up to the EOB of the farthest of those blocks are deleted. The cursor is then placed in the word next to the deleted EOB.

Procedure for deleting blocks

1 Search for or scan a word in the first block of a portion to be deleted.
2 Press the [EOB] key as many times as the number of blocks that you want to delete.
3 Press the editing key [DELETE].

Example of deleting blocks from N01234 to the EOB of a block which is two blocks ahead

1 Search for or scan N01234.

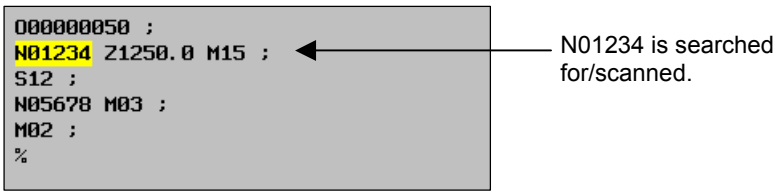


Fig. 10.4.2 (a)

2 Press  .

3 Press the editing key .



Fig. 10.4.2 (b)

10.5 PROGRAM SEARCH

When memory holds multiple programs, a program can be searched for. There are four methods as follows.

Procedure for program search

Method 1

1 Select EDIT or MEMORY mode.
2 Press function key  to display the program screen.
3 Enter a program number or program name. When entering a program number, press address key  then type the program number.
4 Press soft key [PROGRM SEARCH]. Alternatively, press the cursor key  or .
5 Upon completion of search operation, the program number searched for is displayed in the upper-right corner of the screen
If the program is not found, alarm PS0071 occurs.

Method 2

1 Select EDIT or MEMORY mode.
2 Press function key  to display the program screen.
3 Press soft key [PROGRM SEARCH].
4 Then, press soft key [PREV PROGRM] or [NEXT PROGRM].
Pressing [PREV PROGRM] searches for the previous program on the folder, and pressing [NEXT PROGRM] searches for the next program.

Method 3

This method searches for the program number (0001 to 0031) corresponding to a signal on the machine tool side to start automatic operation. Refer to the relevant manual prepared by the machine tool builder for detailed information on operation.

1 Select MEMORY mode.
2 Set the reset state.
- The reset state is the state where the LED for indicating that automatic operation is in progress is off. (Refer to the relevant manual of the machine tool builder.)

3 Set the program number selection signal on the machine tool side to a number from 01 to 31.
4 Press the cycle start button.
- When the signal on the machine tool side represents 00, program number search operation is not performed.
- If the program corresponding to a signal on the machine tool side is not registered, alarm DS0059 is raised.

Method 4 When bit 4 (PGS) of parameter No. 11308 is 1, search for an O number program by specifying digits only.

1 Select EDIT or MEMORY mode.
2 Press function key  to display the program screen.
3 Enter the digits of a program number. Address key  need not be pressed before the digits.
4 Press soft key [PROGRM SEARCH]. Alternatively, press the cursor key  or .

Example
- When bit 4 (PGS) of parameter No. 11308 is 1
  Specify "1000" → Search for program O1000.
  Specify "ABC" → Search for program ABC.
- When bit 4 (PGS) of parameter No. 11308 is 0
  Specify "1000" → Search for program 1000.
  Specify "ABC" → Search for program ABC.

When bit 4 (PGS) of parameter No. 11308 is 1, it is not possible to search for a program with any name consisting of digits only.

10.6 SEQUENCE NUMBER SEARCH

Sequence number search operation is usually used to search for a sequence number in the middle of a program so that execution can be started or restarted at the block of the sequence number.

Example) Sequence number 02346 in a program (O0002) is searched for.

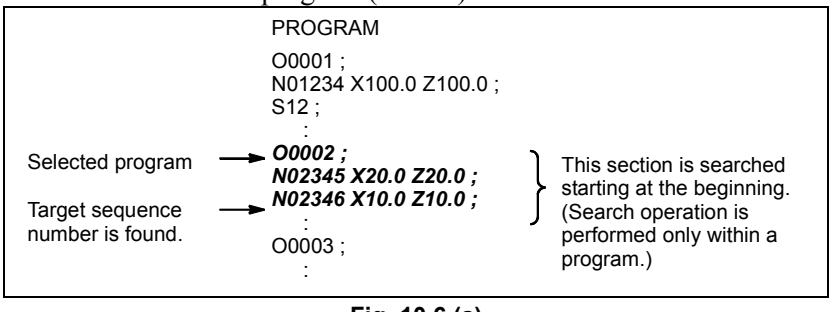


Fig. 10.6 (a)

Procedure for sequence number search

Procedure

1 Select MEMORY mode.
2 Press function key .
3 If the program contains a sequence number to be searched for, perform the operations 4 to 7 below.
If the program does not contain a sequence number to be searched for, select the program number of the program that contains the sequence number to be searched for.
4 Press address key .
5 Key in a sequence number to be searched for.
6 Press soft key [N SRH].
7 Upon completion of search operation, the sequence number searched for is displayed in the upper-right corner of the screen.
If the specified sequence number is not found in the program currently selected, alarm PS0060 occurs.

Explanation

- Operation during Search

Those blocks that are skipped do not affect the CNC. This means that the data in the skipped blocks such as coordinates and M, S, and T codes does not alter the CNC coordinates and modal values.
So, in the first block where execution is to be started or restarted by using a sequence number search command, be sure to enter required M, S, and T codes and coordinates. A block searched for by sequence number search usually represents a point of shifting from one process to another. When a block in the middle of a process must be searched for to restart execution at the block, specify M, S, and T codes, G codes, coordinates, and so forth as required from the MDI after closely checking the machine tool and NC states at that point.

- Checking during search

During search operation, the following checks are made:

- Optional block skip

Limitation

- Searching in sub-program

During sequence number search operation, M98Pxxxx (subprogram call) is not executed. So an alarm PS0060 is raised if an attempt is made to search for a sequence number in a subprogram called by the program currently selected.

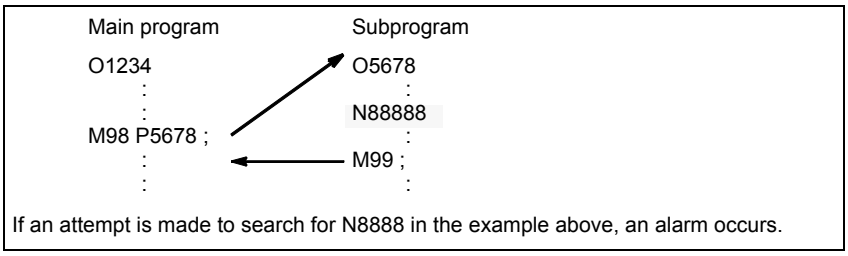


Fig. 10.6 (b)

10.7 DELETING PROGRAMS

Programs registered in memory can be deleted, either one program by one program or all at once.

10.7.1 Deleting One Program

A single program in the folder containing the program currently being edited can be deleted.

Procedure for deleting one program

1 Select the EDIT mode.
2 Press function key  to display the program screen.
3 Press address key .
4 Key in a desired program number.
5 Press the editing key . The program with the entered program number is deleted.

10.7.2 Deleting All Programs

All programs in the folder containing the program currently being edited can be deleted.

Procedure for deleting all programs

1 Select the EDIT mode.
2 Press function key  to display the program screen.
3 Press address key .
4 Key in -9999.
5 Press editing key  to delete all programs.

10.8 EDITING OF CUSTOM MACROS

Unlike ordinary programs, custom macro programs are modified, inserted, or deleted based on editing units.
Custom macro words can be entered in abbreviated form.
Comments can be entered in a program.
Refer to the III-9.1 for the comments of a program.

Explanation

- Editing unit

When editing a custom macro already entered, the user can move the cursor to each editing unit that starts with any of the following characters and symbols:
(a) Address
(b) # located at the start of the left side of a substitution statement
(c) /, (,=, and ;
(d) First character of IF, WHILE, GOTO, END, DO, POPEN, BPRNT, DPRNT and PCLOS
On the screen, a blank is placed before each of the above characters and symbols.
Example) Head positions where the cursor is placed
N001 X-#100 ;

```
#1 =123 ;
N002 /2 X[12/#3] ;
N003 X-SQRT[#3/3*[#4+1]] ;
N004 X-#2 Z#1 ;
N005 #5 =1+2-#10 ;
IF[#1NE0] GOTO10 ;
WHILE[#2LE5] DO1 ;
#[200+#2] =#2*10 ;
#2 =#2+1 ;
END1 ;
```

- Abbreviations of custom macro word

When a custom macro word is altered or inserted, the first two characters or more can replace the entire word.
Namely,

WHILE → WH	GOTO → GO	XOR → XO	AND → AN	SIN → SI	ASIN → AS
COS → CO	ACOS → AC	TAN → TA	ATAN → AT	SQRT → SQ	ABS → AB
BCD → BC	BIN → BI	FIX → FI	FUP → FU	ROUND → RO	END → EN
POPEN → PO	BPRNT → BP	DPRNT → DP	PCLOS → PC	EXP → EX	THEN → TH

Example) Keying in
WH [AB [#2] LE RO [#3]] has the same effect as
WHILE [ABS [#2] LE ROUND [#3]]
The program is also displayed in this way.

10.9 CURSOR MOVEMENT LIMITATIONS ON PROGRAM EDITING

If a program is to be edited while it is stopped or halted, limitations are imposed so that already executed blocks and buffered blocks cannot be edited. Modifiable blocks are those at and below the cursor position at the time the system switches to the edit mode (EDIT). In the remainder of this manual, the area of editable blocks is called the edit-enabled area, while the area of executed blocks and buffered blocks is called the edit-disabled area.
These limitations are applied when the automatic operation status signal <F0000.7> is ON and when a single block execution of MDI operation is edited, and are also applied to TEACH IN HNDL or TEACH IN JOG mode when the playback option is enabled.
This function is enabled when bit 3 (FPD) of parameter No. 11308 is 1.

Explanation

- States where cursor movement limitations are imposed

- EDIT mode while automatic operation is stopped or halted
- MDI mode and when a single block is set
- TEACH IN HNDL or TEACH IN JOG mode (only when the playback function is enabled)

- Cursor movement

Operations that cause the cursor to move to blocks above the edit mode start position are ignored.

- ↑
- ←
- PAGE UP
- CAN（character editing）

- Search and replacement

Operations that cause the cursor to move to the edit-disabled area are disabled.

- Top line search: Results in a search of the top line of the edit-enabled block.
- Specified-line search (character editing): Made through the edit-enabled area.
  If, however, an attempt is made to search through the edit-disabled area, warning "PROGRAM IN OPERATION" is issued, and the operation is disabled.
- Upward search: Made through the edit-enabled area.
  If attempted at the beginning of the edit-enabled block, this operation is ignored.
- Upward search (replacement): Made through the edit-enabled area.
  If attempted at the beginning of the edit-enabled block, this operation is ignored.
- Upward candidate search: Made through the edit-enabled area.
  If attempted at the beginning of the edit-enabled block, this operation is ignored.
- Character string + ↑ (search): Typing the character string to search for and pressing the ↑ key searches for the character string.
  This search is made through the edit-enabled area. If attempted at the beginning of the edit-enabled block, this operation is ignored.
- Replace all: This operation is disabled.

- Reset

The program execution status is canceled, and the cursor movement limitations are also canceled.

- Program search

Program search operations are disabled in following screens.

- Program Editing screen(Word editing/Character editing/Program contents display/Playback)
- Program Check screen
  Warning "PROGRAM IN OPERATION" is issued, and the operation is disabled.
- Program List screen
  When bit 5 (PES) of parameter No. 11302 is 1, only the cursor movement to selected program is executed. Setting to the main program and changing to editing screen are not executed.

- Time insertion

When the machining time stamp function is enabled, soft key [INSERT TIME] does not appear in the state in which the cursor movement limitations are imposed.

10.10 PASSWORD FUNCTION

The password function locks bit 4 (NE9) of parameter No. 3202, which protects programs with program Nos. O9000 to O9999 and programs and folders having the edit/display disable attribute, according to the settings in two parameters, PASSWORD (parameter No. 3210) and KEYWD (parameter No. 3211). In the locked state, parameter NE9 cannot be set to 0. In this case, the protection of programs with program Nos. O9000 to O9999 and programs and folders having the edit/display disable attribute cannot be released unless a correct keyword is set.
A locked state means that the value set in the parameter PASSWD differs from the value set in the parameter KEYWD. The values set in these parameters are not displayed. The locked state is released when the value already set in the parameter PASSWD is also set in parameter KEYWD. When 0 is displayed in parameter PASSWD, parameter PASSWD is not set.

Procedure for locking and unlocking

Locking

1 Set the MDI mode.
2 Enable parameter writing (III-12.3.1). At this time, alarm PS0100 is issued on the CNC.
3 Set parameter No. 3210 (PASSWD). At this time, the locked state is set.

4 Disable parameter writing.

5 Press the  key to release the alarm state.

Unlocking

1 Set the MDI mode.
2 Enable parameter writing (III-12.3.1). At this time, alarm SW0100 is issued on the CNC.
3 In parameter No. 3211 (KEYWD), set the same value as set in parameter No. 3210 (PASSWD) for locking. At this time, the locked state is released.
4 Set bit 4 (NE9) of parameter No.3202 to 0.
5 Disable parameter writing.
6 Press the  key to release the alarm state.
7 Subprograms from program Nos. 9000 to 9999 can now be edited.

Explanation

- Setting parameter PASSWD

The locked state is set when a value is set in the parameter PASSWD. However, note that parameter PASSWD can be set only when the locked state is not set (when PASSWD = 0, or PASSWD = KEYWD). If an attempt is made to set parameter PASSWD in other cases, a warning is given to indicate that writing is disabled. When the locked state is set (when PASSWD ≠ 0 and PASSWD ≠ KEYWD), parameter NE9 is automatically set to 1. If an attempt is made to set NE9 to 0, a warning is given to indicate that writing is disabled.

- Changing parameter PASSWD

Parameter PASSWD can be changed when the locked state is released (when PASSWD = 0, or PASSWD = KEYWD). After step 3 in the procedure for unlocking, a new value can be set in the parameter PASSWD. From that time on, this new value must be set in parameter KEYWD to release the locked state.

- Setting 0 in parameter PASSWD

When 0 is set in the parameter PASSWD, the number 0 is displayed, and the password function is disabled. In other words, the password function can be disabled by either not setting parameter PASSWD at all, or by setting 0 in parameter PASSWD after step 3 of the procedure for unlocking. To ensure that the locked state is not entered, care must be taken not to set a value other than 0 in parameter PASSWD.

- Re-locking

After the locked state has been released, it can be set again by setting a different value in parameter PASSWD, or by turning the power to the NC off then on again to reset parameter KEYWD.

⚠ CAUTION

1 Once the locked state is set, parameter NE9 cannot be set to 0 and parameter PASSWD cannot be changed until the locked state is released or the memory all-clear operation is performed. Special care must be taken in setting parameter PASSWD.
2 The edit/display disable attribute cannot be set unless parameter PASSWD is set.

⚠ CAUTION

3 In the locked state, programs with the edit/display disable attribute are treated as follows:
- The presence of the programs is hidden. This means that these programs are not displayed on screens such as the program folder screen. These programs cannot be edited either.
- These programs cannot be selected as a main program. They can be called as subprograms.

4 In the unlocked state, programs with the edit/display disable attribute are treated in the same manner as ordinary programs.

5 The programs in a folder having the edit/display disable attribute are also treated as described in Caution 3 and 4 above.

6 In the locked state, the folders in a folder having the edit/display disable attribute are treated as follows:
- The presence of the folders is hidden. This means that these folders are not displayed on screens such as the program folder screen.

7 In the unlocked state, the folders in a folder having the edit/display disable attribute are treated in the same manner as ordinary folders.

10.11 EDITING PROGRAM CHARACTERS

This section describes how to edit programs registered in the CNC.
Editing operations include character insertion, modification, deletion, and replacement.
While program word editing is performed by recognizing program words, program character editing is performed on a character-by-character basis.

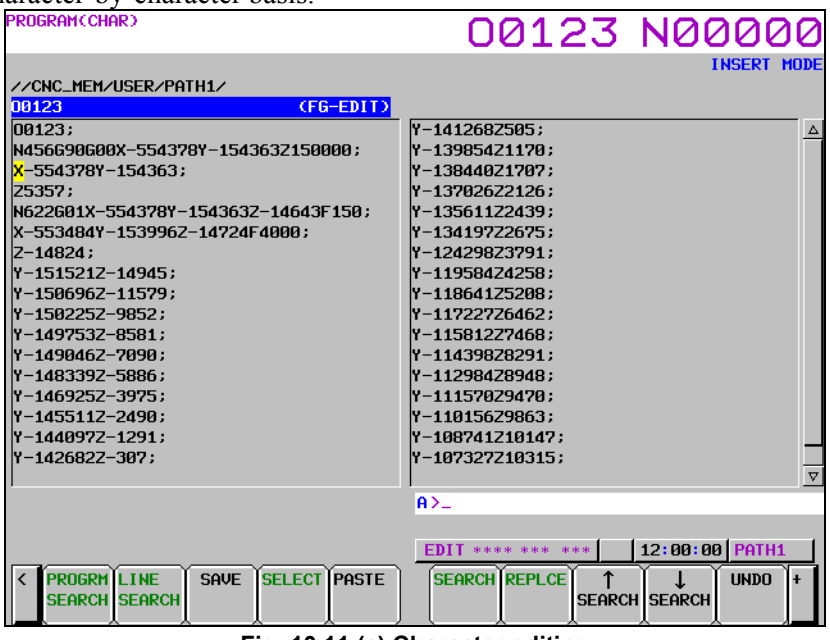


Fig. 10.11 (a) Character editing

Explanation

- Edit unit

Character editing is performed on a character-by-character basis. Select between character editing and word editing according to what to edit.

- Line

A range delimited by an EOB is specified as a line.
In character editing, edited data is saved on a line-by-line basis.
When one program line contains many characters, it extends over multiple lines on the screen, but these lines are counted as one program line.

- Line splitting

When the cursor is on a line during line editing, pressing edit key 

causes the line to be split into two, one before the cursor and the other after the cursor.

Pressing edit key 

immediately after the line is split causes the lines to return to a single one.

Alternatively, moving the cursor to the EOB of the first line resulting from the split and pressing edit key 

causes the lines to return to a single one.

- Line merging

If the EOB of a line is deleted, the line is combined with the next line to form a single line.

- Maximum number of characters in one line

One line can consist of up to 140 characters.

- Line number

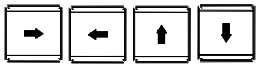
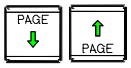
The number of lines are counted starting with the starting line of a program, which is counted as the first line. Even when a line wraps around to the next and subsequent lines, these lines are counted as a single line.

- Clipboard

The clipboard is an area to store characters when a cut or copy operation is performed. The area has a capacity for holding approximately 4000 characters.
The characters that are cut or copied by one cut or copy operation are stored. The most recently stored character string can be used by the paste function.
The information stored by the paste operation is kept unchanged until another cut or copy operation is performed.
The characters stored in the clipboard are maintained until the power to the CNC is turned off unless the clipboard is updated again.

- Undo function

The function for undoing operations performed in program editing undoes operations sequentially, starting with the one performed last. It can undo only those operations that update character strings.
A single undo operation undoes a single input or deletion operation.
If data updated with replace operations is to be undone, a single undo operation undoes the replacement of a single item.

Operations of the cursor keys  and the page keys 

cannot be undone with this function because they do not update character strings.

- Example of the undo function

1 Suppose that the following character string is present before modifications:
N110AX[#AXIS3]=100.0;

2 Edit key 

is pressed five times.
X[#AXIS3]=100.0;

3 Soft key [UNDO] is pressed five times. Then, the original character string is restored as follows:
AX[#AXIS3]=100.0;

0AX[#AXIS3]=100.0;
10AX[#AXIS3]=100.0;
110AX[#AXIS3]=100.0;
N110AX[#AXIS3]=100.0;

- Input modes for editing

Input modes for program editing include insert mode and overwrite mode.
To switch between the input modes, press soft key [INPUT MODE].
Initially, insert mode is set.

- Insert mode
  In insert mode, an entered character is inserted between the current cursor position and the preceding character position.
  1234567890
  When the cursor is placed at 6, and X is entered, the following results:
  12345X67890

- Overwrite mode
  In overwrite mode, the character at the cursor position is replaced by an entered character.
  1234567890
  When the cursor is placed at 6, and X is entered, the following results:
  12345X7890

- Restrictions on editing

O numbers and file names cannot be edited.
EOR (%) cannot be deleted.

- Line editing and automatic saving

When a line is edited, the line is displayed in the update color, blue (which can be changed with color settings), to indicate that the line has been edited. When the line is displayed in the update color, the edit has not yet been saved.
When the cursor is moved to a line preceding or succeeding the edited line, the edit is automatically saved, and the line display changes from the update color to the normal color.
Alternatively, pressing soft key [SAVE] enables the edit to be saved.

- Function for automatically saving an edited line vs. function for undoing editing operations (UNDO function)

When an edit operation is undone, the line is displayed in the update color, and the edit is not saved any longer. If the cursor moves to another line when the operation is undone, the edit is automatically saved.

10.11.1 Available Keys

The available keys are as follows:

- Cursor keys

Cursor keys , , , and  move the cursor.

- Editing key 

Deletes the character at the cursor position.

- **Editing key** 

Deletes the character immediately before the cursor position.
When the cursor is at the beginning of a line, the character at the end of the previous line is deleted.

- **Editing key** 

Causes a line change.

- **Page change keys**

Pages are changed using page key 

or 

Page key 

moves to the next page.

When page key 

is pressed on the last page, the cursor moves to the last character position of the last line (the % position).

Page key 

moves to the previous page.

When page key 

is pressed on the first page, the cursor moves to the first character position of the first line.

- **Character keys**

Characters are entered using these keys.

10.11.2 Input Mode

Input modes include insert mode and overwrite mode.

Changing input mode

To switch between the input modes, use soft key [INPUT MODE].
Pressing soft key [INPUT MODE] causes insert mode and overwrite mode to be selected alternately.
The current mode is indicated in the lower right part of the editing screen.

10.11.3 Line Number Display

This function is used to display a program with line numbers. Pressing soft key [LINE NUMBER] displays a program with line numbers. Pressing soft key [LINE NUMBER] again causes the line numbers to disappear.

10.11.4 Search

A program is searched for a character string.

Search

Procedure

1 Press soft key [SEARCH].
2 A search character string input area is displayed. Enter the character string to search for. To cancel the search, press soft key [CANCEL].
3 Upward search operation
Pressing soft key [UP] causes the program to be searched in the upward direction (toward the top), starting at the current cursor position.

When a search character string is found, the cursor moves to the character string.
Pressing soft key [UP] again causes the program to be searched for the next candidate.

4 Downward search operation
Pressing soft key [DOWN] causes the program to be searched in the downward direction (toward the bottom), starting at the current cursor position.
When a search character string is found, the cursor moves to the character string.
Pressing soft key [DOWN] again causes the program to be searched for the next candidate.

10.11.5 Replacement

A character string in a program is replaced with a specified character string.

Replacement

Procedure

1 Press soft key [REPLCE].
2 Search and replacement character string input areas are displayed. Enter a search character string (character string to be replaced) and a replacement character string (character string to replace the former character string).
To move between the search character string input area and the replacement character string input area, use the cursor keys  and .
If a search character string is input, but no replacement character string is input, the former character string will be deleted.
To cancel the replacement, press soft key [CANCEL].
3 Replacement operations
Replacement operations include an operation for searching for a search character string and an operation for actually replacing it.
To perform an operation for searching for a search string, use soft key [↑ SEARCH] or [↓ SEARCH].
To perform an operation for actually replacing it, use [REPLCE] or [REPLCE ALL].

- Soft key [↑ SEARCH]
  Searches the program in the upward direction (toward the top), starting at the current cursor position.
  When a search character string is found, the cursor moves to the character string.
  Pressing soft key [↑ SEARCH] again causes the program to be searched for the next candidate.
- Soft key [↓ SEARCH]
  Searches the program in the downward direction (toward the bottom), starting at the current cursor position.
  When a search character string is found, the cursor moves to the character string.
  Pressing soft key [↓ SEARCH] again causes the program to be searched for the next candidate.
- Soft key [REPLCE]
  Replaces the character string found with a search operation with a replacement character string.
- Soft key [REPLCE ALL]
  Replace all instances of the search character string in the program text with the replacement character string.
  Pressing this soft key causes message "ARE YOU SURE YOU WANT TO EXECUTE?" to appear and also soft key [YES] and soft key [NO] to appear. Pressing soft key [YES] causes all replacements to be executed.
  Pressing soft key [NO] cancels the replacement operation, without executing the replacements.

10.11.6 Reversing Edit Operations (Undo Function)

Edit operations performed on a program can be undone sequentially starting with the one performed last.

Reversing edit operations (undo function)

Procedure

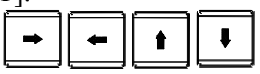
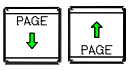
1 Pressing soft key [UNDO] once undoes a single operation.
When there are no longer any operation to be undone, pressing this soft key has no effect.

10.11.7 Copy

A selected character string can be copied to the clipboard.

Copying

Procedure

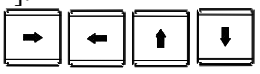
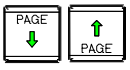
1 Move the cursor to the beginning of the character string to select.
2 Press soft key [SELECT].
3 Using the cursor keys  and the page keys , move the cursor to the end of the character string to select.
At this time, the background of the selected character string is displayed in the selection color (cursor color).
4 Press soft key [COPY] to copy the selected character string to the clipboard.
To cancel the copying, press soft key [CANCEL].

10.11.8 Cut

A selected character string can be deleted.
When deleted, the character string is copied to the clipboard.

Cut

Procedure

1 Move the cursor to the beginning of the character string to select.
2 Press soft key [SELECT].
3 Using the cursor keys  and the page keys , move the cursor to the end of the character string to select.
At this time, the background of the selected character string is displayed in the selection color (cursor color).
4 Press soft key [CUT] to delete the selected character string.
To cancel the deletion, pres soft key [CANCEL].

10.11.9 Paste

The character string stored in the clipboard can be inserted at the current cursor position.
After pasted, the character string remains stored in the clipboard, so that the character string can be pasted as many times as necessary.

Paste

Procedure

1 Move the cursor to the position at which you want to paste a character string.
2 Press the soft key [PASTE].

10.11.10 Saving

Moving the cursor to a line preceding or succeeding the edited line causes the edit to be automatically saved, but it is possible to save the edit by pressing soft key [SAVE], if necessary. For a program consisting of only a single line, in particular, it is necessary to press soft key [SAVE] to save the edit.

Saving

Procedure

1 Press the soft key [SAVE].

10.11.11 Creation

A new program can be created.

Creation

Procedure

1 Press soft key [NEW PROGRM].
2 A program name input area appears.
3 Enter the name of a program to be created.
4 Press the soft key [EXECUTE]. This creates a new program and displays the editing screen.
To cancel the creation of a new program, press soft key [CANCEL].

10.11.12 Line Search

The cursor can be moved to a specified line.
The cursor can be moved to a line with a specified line number, to the first line of the program, and to the last line of the program.

Line search

Procedure

Movement to a line with a specified line number

1 Press soft key [LINE SEARCH].
2 A line number input area appears.
3 Enter a line number. The first line is regarded as having a line number of 1.
4 Press soft key [LINE NUMBER], and the cursor moves to the specified line.

Movement to the first line

1 Press soft key [LINE SEARCH].
2 Press soft key [TOP], and the cursor moves to the first line of the program.

Movement to the last line

1 Press soft key [LINE SEARCH].
2 Press soft key [BOTTOM], and the cursor moves to the last line of the program.

10.12 PROGRAM COPY FUNCTION

A program is copied or moved between folders.

Procedure for copying a program compact

Procedure

1 Press function key 

.

2 Press chapter selection soft key [FOLDER].
The program folder screen in the Fig. 10.12 (a) appears:

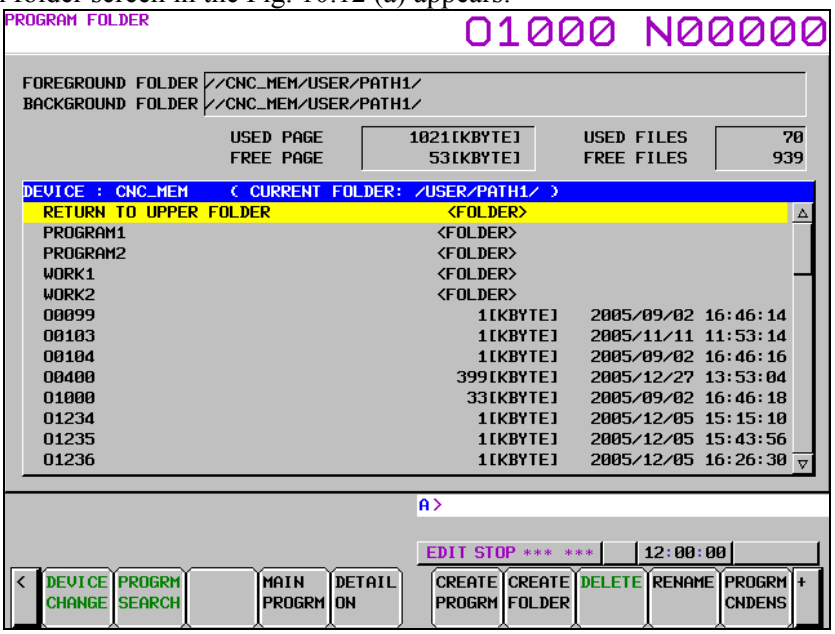


Fig. 10.12 (a)

3 Press the soft key [(OPRT)].

4 Move the cursor to a folder that contains the program that you want to copy or move, and press 

key.

5 Press the continuous menu key  until soft key [SELECT START] appears.
Press the soft key [SELECT START].

6 Move the cursor to the program that you want to select, and press soft key [SELECT]. The background color of the selected program becomes light blue, indicating the selected state.

7 Press the soft key [SELECT END]. The selected program is determined.

8 Press 

key, and move the cursor to the folder to which you want to copy or move the program.

9 Press the soft key [COPY]. The selected program is copied. If just one program is selected, pressing soft key [COPY] after typing a program name performs a copy operation with the entered name.

10 To move the program, press soft key [MOVE]. The selected file is then moved. If just one program is selected, pressing soft key [MOVE] after typing a program name performs a move operation with the entered name.

Explanation

Operations are accepted only when the data protection key is set to ON.

If the program storage capacity on the copy destination side is insufficient, the copy operation is not accepted.

The currently selected program is highlighted. Multiple programs can be selected in the same folder. Each time [SELECT] is pressed, the program currently indicated by the cursor is selected.

A selected program can be deselected by pressing [SELECT] again or by pressing [CANCEL].

If the same program name is already present in the copy or move destination folder, "REWRITE: FILE NAME" is displayed, allowing you to determine whether to overwrite the existing program by pressing soft key [NO] or [REWRITE]. Pressing [REWRITE] performs overwriting, and pressing [NO] does not perform the copy or move operation of that program.

Pressing soft key [DELETE] deletes all the files currently selected.

A program can be neither copied nor moved to the same folder as the selected folder.
When only one program is selected, and a program name is already entered, however, the program can be copied or moved within the same folder.

NOTE
Once a copy or move operation starts, it cannot be canceled. So, start the operation very carefully.

10.12.1 Copying and Moving Files between Devices

Overview

Files can be copied and moved between different devices.
There are two procedures available on the program folder screen.

<1> File copying and moving (device change)
Select a source file first, and then select a destination folder by first changing the device and then selecting a folder, decide on the destination folder, and execute copy or move.

<2> File copying and moving with split display
On 10.4-/15-/19-inch display units, the program folder screen can be split. By using this screen split function, it is possible to perform copy or move operations while checking the file to be copied or moved on the screen on which the folder containing the source file and the destination folder are displayed at the same time.

File copying and moving (device change)

Procedure

1 Select EDIT mode.
2 Press function key 

to display the program folder screen.
3 Press soft key [(OPRT)].
4 Press the continuous menu key  until [SELECT START] appears. Press soft key [SELECT START].
5 Use the cursor keys as appropriate to position the cursor on the desired file.
6 Press soft key [SELECT] to select the file.
7 After the desired file has been selected, press soft key [SELECT END].
8 To select a destination device, press soft key [DEVICE CNAHGE].
9 Select the desired device, using the soft key.
10 In the selected device, move to a destination folder.
11 To copy the selected file to the destination folder, press soft key [COPY], and to move the file to the folder, press soft key [MOVE].

File copying and moving with split display

Procedure1

- Procedure for selecting a file first and then a destination folder

1 Select EDIT mode.
2 Press function key  to display the program folder screen.
3 Press soft key [(OPRT)].
4 Press the continuous menu key  until [MULTI LIST] appears. Press soft key [MULTI LIST].
5 In either of the folder displays, select the folder containing the source file.
6 Press soft key [SELECT START].
7 Use the cursor keys as appropriate to position the cursor on the desired file.
8 Press soft key [SELECT] to select the file.
9 Use the  + cursor key combination, select the other folder display.
10 To select a destination device, press soft key [DEVICE CNAHGE].
11 Select the desired device, using the soft key.
12 In the selected device, move to a destination folder.
13 To copy the selected file to the destination folder, press soft key [COPY], and to move the file to the folder, press soft key [MOVE].

Procedure 2

- Procedure for selecting a destination folder first and then a file

1 Select EDIT mode.
2 Press function key  to display the program folder screen.
3 Press soft key [(OPRT)].
4 Press the continuous menu key  until [MULTI LIST] appears. Press soft key［MULTI LIST］.
5 In either of the folder displays, select the folder containing the source file, use the  + cursor key combination to switch to the other folder display, and on this folder display, select a destination folder.
6 Use the  + cursor key combination to switch to the folder display in which the source folder has been selected.
7 Press the continuous menu key  until [SELECT START] appears.
Press soft key [SELECT START].
8 Use the cursor keys as appropriate to position the cursor on the desired file.
9 Press soft key [SELECT] to select the file.
10 After the file has been selected, press soft key [COPY] to copy the selected file to the folder selected in the other folder display and press soft key [MOVE] to move the file.

Explanation

- Multiple selectable files

Only multiple files that are in the same folder can be selected at the same time with soft key [SELECT]. Files in other devices and folders cannot be selected at the same time.
Up to ten files can be selected at the same time.

- Changing the file name during copying and movement

If only one file is selected for copying or moving, the file can be copied or moved by changing its file name.
After selecting a file, enter a file name and press soft key [COPY] or [MOVE]. This causes the file to be copied or moved with the entered file name.

- Copying and moving a file to the same folder

It is not possible to copy and move a selected file to the folder that contains the file.
If an attempt is made to do so, warning "SAMEFILE NOTCOPY" is issued.
If, however, only one file is selected, and is to be copied or moved by changing its file name, it can be copied or moved to the same folder.

- Overwriting files

When copying or moving is attempted, if a file with the same name exists in the destination folder, message "REWRITE: File name" flashes below the key-in buffer. If the file with the same name is to be overwritten, press soft key [REWRITE] and otherwise, press soft key [NO].

- File search when selecting a file

After soft key [SELECT START] is pressed, it is possible to search for a file through a folder using soft key [SEARCH] when selecting the file to copy or move.
Enter the file name to search for first and then press soft key [SEARCH]. If a file with the entered file name exists, the cursor is positioned on the file. If no file with the entered file name exists, warning "SPECIFIED PROGRAM NOT FOUND" appears.
Soft key [SEARCH] simply moves the cursor to the program found on the directory screen. It is not possible to switch operations, using bit 5 (PES) of parameter No. 11302.

- Display of a selected file name

When a file is selected with soft key [SELECT], the color of the background of the file name display is changed. The color of the background of the file name display remains unchanged until the file is deselected.

- Deselecting a file

When a file is selected with soft key [SELECT], the selected file can be deselected by positioning the cursor on the file and pressing soft key [SELECT] again. All selected files are deselected when any of the following operations is performed.

- Pressing of soft key [CANCEL] after performing a select operation.
- Completion of a move operation.
- Change of the source folder during selection.
- Termination of a select operation with soft key [SELECT END] and pressing of soft key [SELECT START] to enter the start of selection state.

NOTE
When bit 6 (RTC) of parameter No. 11318 is 1, all selected files are also deselected upon completion of a copy operation.

Limitation

- Mode for performing copy and move operations

To copy and move files between different devices or data servers, enter EDIT mode. In a multi-path system, place all paths in EDIT mode. If an attempt to perform an operation in a mode other than EDIT mode, warning "CHANGE TO EDIT MODE" is issued.

- Restrictions on file operations in various devices

Those devices between which file copy and move operations can be performed are as follows:

- CNC MEM
- MEM CARD (binary format)
  (Note that a folder on a MEM CARD can be selected as a source folder from which a file is to be read.)
- Data server

Between devices other than those above, copy or move operations cannot be performed.

It is not possible to copy and move to devices other than those above or to a MEM CARD or to move from a MEM CARD.
If an attempt is made to copy or move, warning "CAN NOT COPY/MOVE" is issued.

- Folder copy and move operations

Folder copy or move operations cannot be performed.
If an attempt is made to select a folder as an object to copy or move, warning "DISABLE TO SELECT" is issued.

Note

NOTE
Once a copy or move operation starts, it cannot be canceled. Thus, check the files and folder subject to the operation carefully before starting it.

10.13 KEYS AND PROGRAM ENCRYPTION

Overview

Program contents can be protected by setting parameters for encryption and for the program security range.

Explanation

1 Security with a password and a security range
When the password and security range parameters are specified, the display, editing, and input/output operations are disabled for the programs within the security range.
This prevents the custom macro programs created by the machine tool builder from being accidentally changed or deleted by the end user. This also provides security because the contents of the programs are not displayed.

2 Input/output of encrypted programs
The programs in the security range can be encrypted before being output. Once encrypted, the programs cannot be decrypted. In addition, encrypted programs can be input directly.

- Locking/unlocking

When the programs in the security range are secured, the program memory is said to be locked.
If they are not secured, the program memory is said to be unlocked.

- Parameter

- Parameter PASSWORD (No.3220)
  Set the password required to lock the program memory. A password other than 0 can be set. The password is not displayed. A password can be set when no password is set (PASSWORD = 0) or the program memory is unlocked.
- Parameter KEY (No.3221)
  When KEY is set equal to PASSWORD, the program memory is unlocked. The value set for the parameter is never displayed. Each time the CNC is started, 0 is set for the parameter. This means that whenever the CNC is started, a lock is provided if a password is set (PASSWORD $\neq$ 0).
- Minimum value (in parameter No. 3222) and maximum value (in parameter No. 3223) of the program security range
  Set the desired security range. The value set for MINIMUM must not be greater than that set for MAXIMUM. The programs in the range of MINIMUM to MAXIMUM are secured. If MINIMUM is set to 0, it is regarded as being 9000. If MAXIMUM is set to 0, it is regarded as being 9999.
  A range can be set when no password is set or the program memory is unlocked.

NOTE
1 For security, the values set for PASSWORD and KEY are not displayed. For the same reason, PASSWORD, MINIMUM, and MAXIMUM can be specified only when no password is set or the program memory is unlocked. Set a password, taking great care to avoid a situation where the program memory cannot be unlocked because a password is set incorrectly.
2 The [+INPUT] key used to specify PASSWORD and KEY behave in the same way as the [INPUT] soft key.

Example: When 99 is set for KEY, pressing [1] and [+INPUT] changes the value to 1.

3 These four parameters cannot be output externally. Also, even when these parameters are input by the parameter input operation, they are ignored.

- Inputting/outputting and collating programs

When a program is encrypted, a password is output. The password is used to load the program.
The following output operations are enabled for programs outside the security range if the locked state is present or for programs within the security range if the unlocked state is present:

Outputting all programs

Locked/unlocked	Results
Locked	All the programs outside the security range are output in the normal way.
Unlocked	All the programs within the security range are encrypted and output.
Password not set	All the programs in the program memory are output in the normal way.

Outputting a single program

Locked/unlocked	Results
Locked	If a program is outside the security range, it is output in the normal way. If it is within the security range, a warning "PROTECTED" is issued.
Unlocked	If a program is outside the security range, it is output in the normal way. If it is within the security range, it is encrypted and output.
Password not set	The program is output in a normal way.

Outputting specified multiple programs

Locked/unlocked	Results
Locked	When all of the specified programs fall outside the protected range, they are output as usual. When all of the specified programs are within the security range, warning message "PROTECTED" is issued. When some of the specified program numbers are outside the security range, and the others are within the security range, only the programs outside the security range are output normally. When the programs to be output are not within the specified range, warning message "THE WRONG DATA IS USED" is issued.
Unlocked	When all of the specified programs fall outside the protected range, they are output as usual. When all the specified programs fall within the protected range, they are output as encoded programs. When some of the specified program numbers are outside the security range, and the others are within the security range, only the programs within the security range are encoded and output. When the programs to be output are not within the specified range, warning message "THE WRONG DATA IS USED" is issued.

Locked/unlocked	Results
Password not set	The program is output in a normal way.

Inputting an un-encrypted program

Locked/unlocked	Results
Locked	When the program to be input is outside the security range, it is input normally. When the program to be input is within the security range, warning message "PROTECTED" is issued.
Unlocked, or password not set	The program is input.

Inputting an encrypted program

Password set in the system and password of the program	Results
Password set in the system	Warning “WRITE PROTECT” is issued.
Password for the program	When the program is within the security range, it is input normally. When the program is outside the security range, warning message "WRITE PROTECTED" is issued.
Password not set in the system	The program is input. The PSW in the file is set for parameter No. 3220. This applies if the program is in the security range. If the program is outside the security range, warning message "PROTECTED" is issued.

Collating a program with an encrypted program

In the unlocked state, the following takes place:

Password set in the system and password of the program	Results
Password set in the system	Alarm SR0075 “PROTECT” is issued.
Password set in the system = Password of the program, or password not set in the system	The program is collated.

In the locked state, program collation cannot be performed.

NOTE
To encrypt programs, set parameter ISO (bit 1 of No.0000) to 1 (to specify that the output code is ISO).

- Program display

1 On the program folder screen, all program numbers are displayed together with comments.
2 In the locked state, the programs within the security range are not displayed on the program screen. In the unlocked state, the programs within the security range are displayed in the same way as normal programs.

NOTE
Immediately after switched from the unlocked state to the locked state or from the locked state to the unlocked state, programs are displayed in the previous state. They are displayed in the new state when a program search or other operation is performed.

- Editing and deleting programs

When the program memory is locked, the programs within the security range cannot be edited or deleted. When the program memory is locked, an attempt to delete all programs results in only those programs outside the security range being deleted.
In the locked state, the programs in the security range cannot be selected as a main program. If an attempt is made to select such a program on the program folder screen as a main program, warning message "SETTING A MAIN PROGRAM FAILED" is issued.

- Searching for programs

In the locked state, a program search is performed within the protected range as described below.

1 When no program number is specified, programs within the protected range are skipped.
2 When an attempt is made to search for a program within the security range by specifying a program number, the search operation is ignored, resulting in warning message "Program is protected".

10.14 SIMULTANEOUS EDITING OF MULTIPATH PROGRAMS

Simultaneous editing of multi-path programs enables simultaneous editing of programs for multiple paths on a single screen.
This function is enabled when bit 0 (DHD) of parameter No. 3106 is 1.

Explanation

- Procedure

1 Set the paths subject to simultaneous editing, with multi-path simultaneous display parameters Nos. 13131 and 13132.
(For details of the parameters, refer to the Parameter Manual (B-64490EN).)
2 Place the paths subject to simultaneous editing in EDIT mode.
3 Press function key 

.
4 Press soft key [PROGRAM] to display the program editing screen.

- Screen display

Fig. 10.14 (a) and Fig. 10.14 (b) show examples of performing simultaneous editing of multi-path programs.
Above each program is a status line on which three items of information are displayed: the program name, "FG-EDIT", which indicates that the program is being edited in the foreground, and the path name.
For the program currently being edited, its status line is displayed in reverse video.

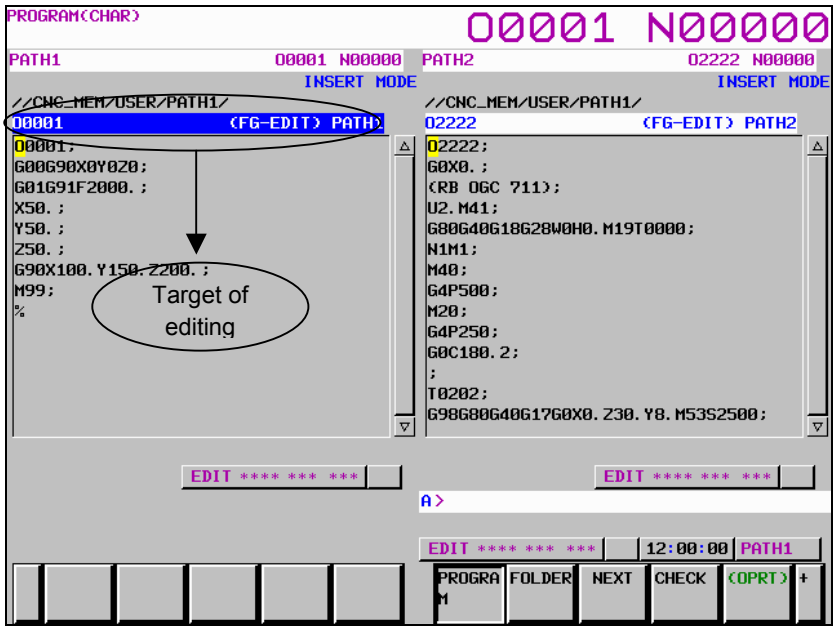


Fig. 10.14 (a) Simultaneous editing of multi-path programs screen (10.4-inch display unit)

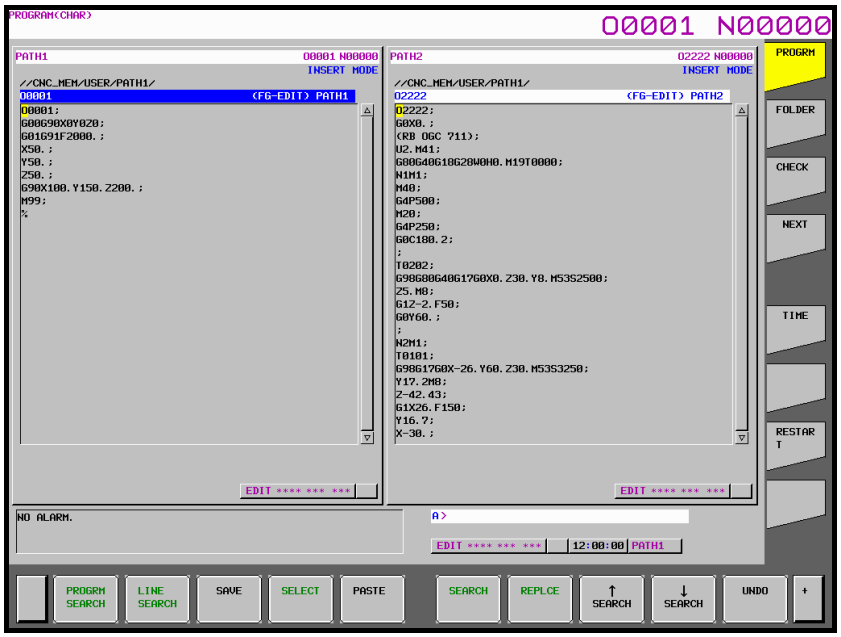


Fig. 10.14 (b) Simultaneous editing of multi-path programs screen (15-inch display unit)

- Modes

When the paths to be displayed simultaneously are in either EDIT or MEM mode, the multi-path programs are displayed simultaneously on the program screen. Selecting the path for which EDIT mode is selected enables program editing to be performed.

Fig. 10.14 (c) shows an example in which both EDIT and MEM modes are selected.

For path 1, MEM mode (execution status screen) is selected, and for path 2, EDIT mode (editing screen) is selected.

Soft keys are switched according to the mode for the currently selected path.

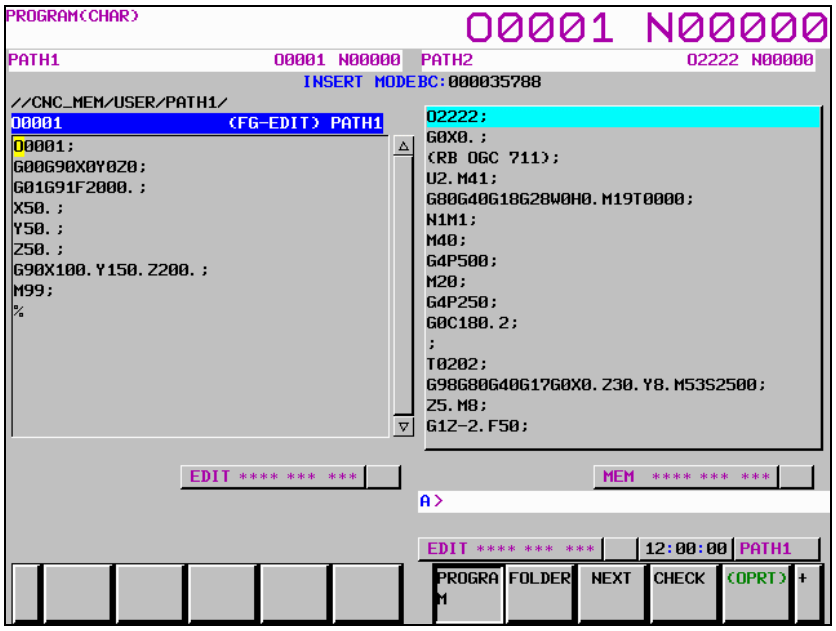


Fig. 10.14 (c) Screen on which both MEM and EDIT modes are selected

- Switching the path subject to editing

The path selected with the path selection signal is subject to editing.

- Maximum number of paths that can be subject to editing simultaneously

The maximum number of paths that can be subject to editing simultaneously on each display unit is as given in the Table 10.14 (a).

Table 10.14 (a)

Display unit size	Maximum number of paths that can be subject to editing simultaneously
8.4 or 10.4-inch	3
15 or 19-inch	4

- Conditions under which simultaneous editing is not usable

Simultaneous editing of multi-path programs is disabled in the following cases:

- The program screen is selected as a small screen.
- The paths to be displayed simultaneously include a path for which a mode other than EDIT and MEM modes is selected.
- The virtual MDI key function is enabled.

If, in the state of simultaneous editing of multi-path programs, background editing is started, background editing is performed on all screens. Simultaneous editing of multi-path programs and background editing cannot be used at the same time.

- Simultaneous editing on 8.4-inch display unit

When simultaneous editing is performed on 8.4-inch display unit, the characters get smaller.
The number of characters per path in the editing area is as follows:

- Display of 38 columns and 10 rows for non-simultaneous display
- Display of 74 columns and 15 rows for 1-path simultaneous display
- Display of 35 columns and 14 rows for 2-path simultaneous display
- Display of 22 columns and 14 rows for 3-path simultaneous display

10.15 MULTI-PATH EDITING FUNCTION

10.15.1 Overview

When the program of the path to be edited is scrolled in the simultaneous editing of multi-path programs screen, other path programs that are displayed on the same screen can be scrolled simultaneously.

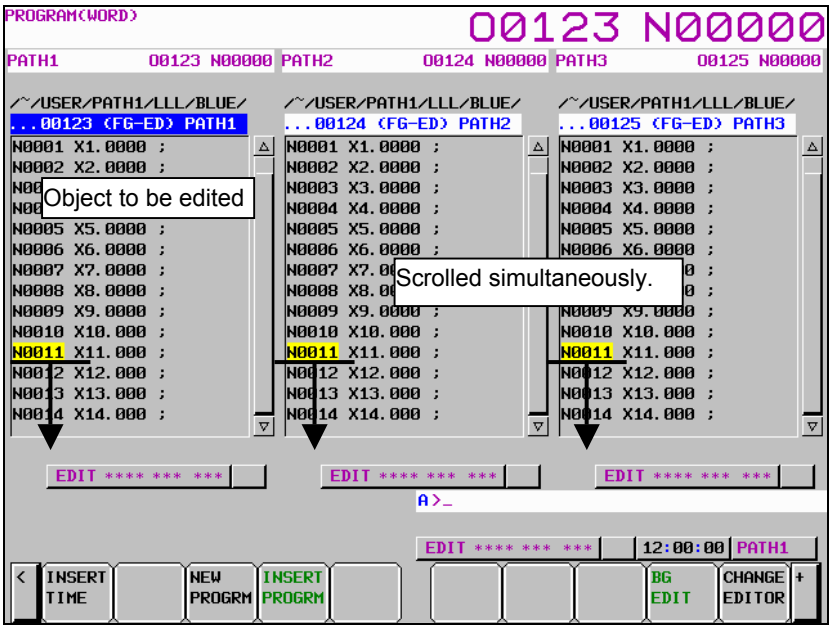


Fig. 10.15.1 (a) Synchronous scroll function

This function provides the simultaneous scroll mode in which all programs being edited simultaneously are scrolled and the single scroll mode in which only the program to be edited is scrolled. It is possible to switch between these modes easily through soft key operation.

In the simultaneous scroll mode, when a waiting M-code comes due to a scroll, waiting is performed by stopping the scroll of the path until the arrival of the same waiting M-code of other paths. It is, therefore, possible to edit the program while confirming the waiting of each path.
Moreover, the waiting search function enables the cursor positions of all paths being edited simultaneously to be moved to specified waiting M-code simultaneously.

NOTE
This function is an optional function.

10.15.2 Details

Switching between simultaneous and single scroll modes

The following conditions are necessary for switching between simultaneous scroll mode and single scroll mode on the program editing screen.

- Bit 0 (DHD) of parameter No. 3106 for enabling the simultaneous multi-path program editing function is 1.
- There are two or more paths in the simultaneous editing mode.
- All paths being edited simultaneously are in EDIT mode.
- All program editing screens for simultaneous editing are in word editor mode.

NOTE
1 The single scroll mode is selected when the power is turn on.
2 If the above conditions are not satisfied, the scroll mode is automatically switched to single scroll mode.

Procedure for switching to the simultaneous scroll mode

The procedure for switching to the simultaneous scroll mode is as described below.

1 Press function key .
2 Press soft key [PROGRAM] to display the program editing screen.
3 Press soft key [(OPRT)].
4 Press continuous menu key [+] several times until oft key [SYNC SCROLL] appears.
5 Press soft key [SYNC SCROLL].
6 The screen changes to the simultaneous scroll mode. At this time, the display of the soft key [SYNC SCROLL] changes to [SINGLE SCROLL].

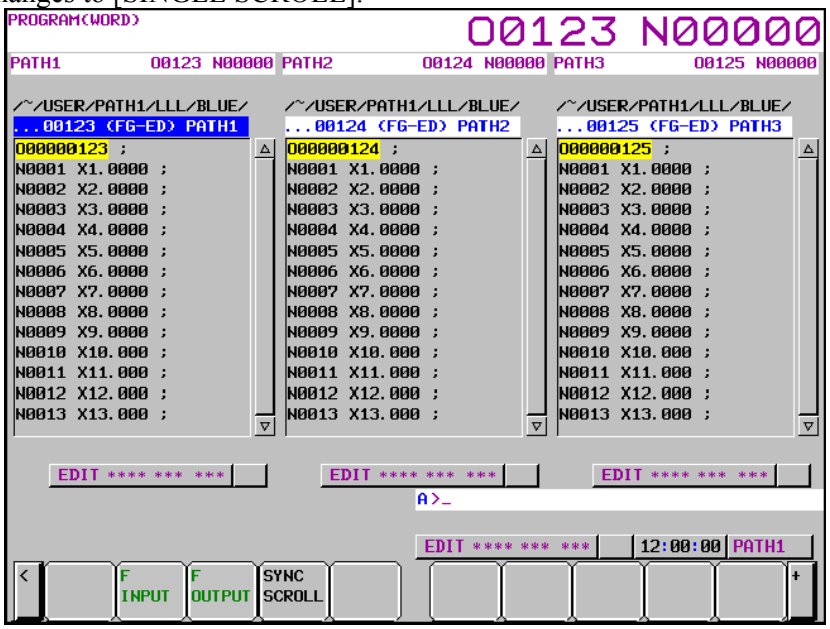


Fig. 10.15.2 (a) Multi-path editing screen

Procedure for switching to the single scroll mode

The procedure for switching to the single scroll mode is as described below.

1 Press function key .
2 Press soft key [PROGRAM] to display the program editing screen.
3 Press soft key [(OPRT)].
4 Press continuous menu key [+] several times until oft key [SINGLE SCROLL] appears.
5 Press soft key [SINGLE SCROLL].
6. The screen changes to the single scroll mode. At this time, the display of the soft key [SINGLE SCROLL] changes to [SYNC SCROLL].

Cursor movement operations

In the simultaneous scroll mode, pressing the cursor movement keys  causes the cursors to move up and down in all programs being edited simultaneously.

If the cursor movement keys  are pressed to move within the same block, the cursors for the paths other than the one to be edited do not move.

If the cursor movement keys [←] [→] are pressed to move the cursor to the previous and next blocks, the cursors for the paths not to be edited also move.
Those programs for paths that are not displayed on the screen do not scroll simultaneously.

Page change operation

In the simultaneous scroll mode, pressing the page change keys [PAGE ↓] [PAGE ↑] causes the page to change in all programs being edited simultaneously.

Scroll waiting with a waiting M-code

In the simultaneous scroll mode, if the cursor moves to a block that contains a waiting M-code, the system enters the scroll waiting state.
In the scroll waiting state, the cursor cannot be moved in the direction in which the cursor has moved to the block that contains a waiting M-code until the cursors of all programs subject to waiting move to the same waiting M-code.

In the scroll waiting state, the cursor color changes to blue as shown in Fig. 10.15.2 (b).

NOTE
A waiting M code is not ignored due to the waiting ignore signal.

Example:

The cursor cannot be moved in the down direction if pressing the cursor key [↓] causes the system to enter the scroll waiting state. The cursor can move in the up direction.

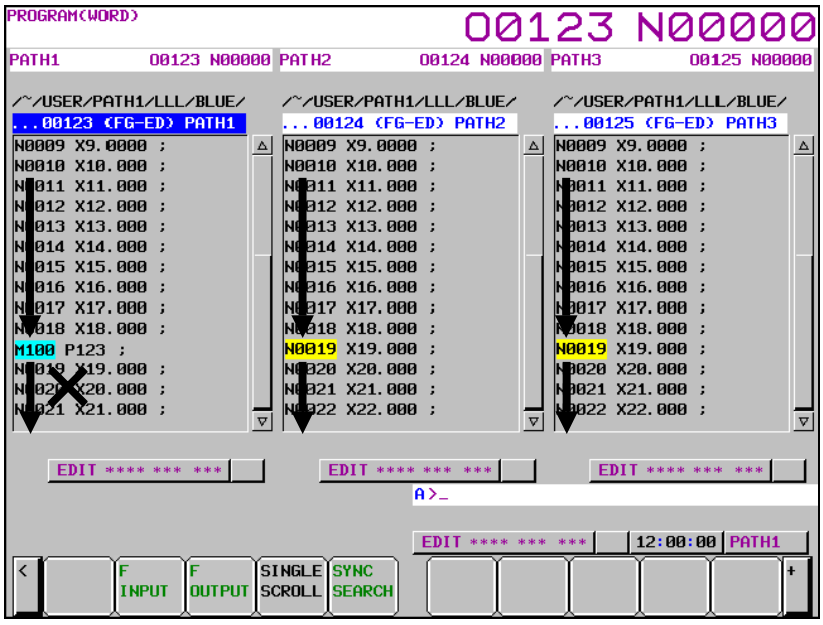


Fig. 10.15.2 (b) Waiting caused by pressing the cursor key

Similarly, if the cursor is moved to a block that contains a waiting M-code by pressing a page change key, the system enters the scroll waiting state in the block that contains a waiting M-code.

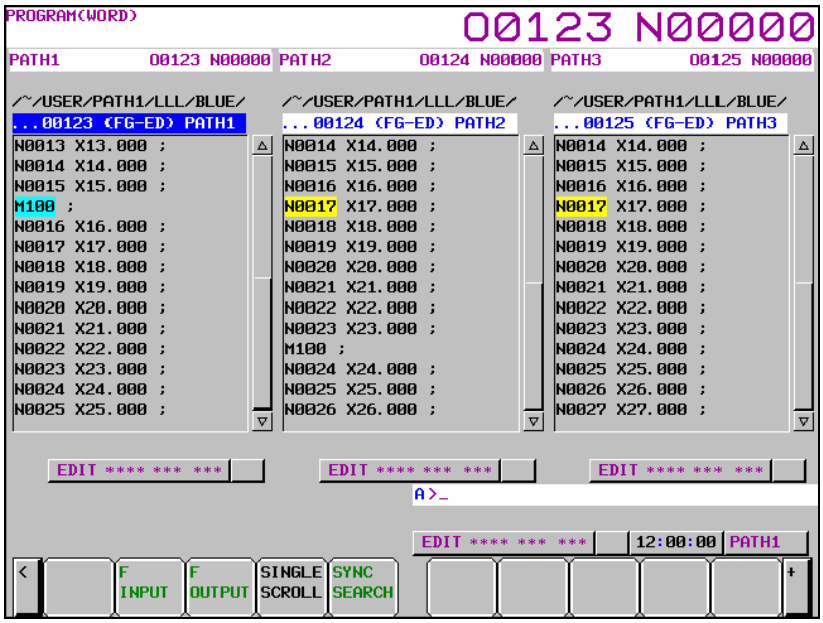


Fig. 10.15.2 (c) Scroll waiting caused by pressing a page change key

Completion of scroll waiting

When the cursors move to the same waiting M-code in all programs subject to waiting, scroll waiting is completed, so that scrolling can be continued.

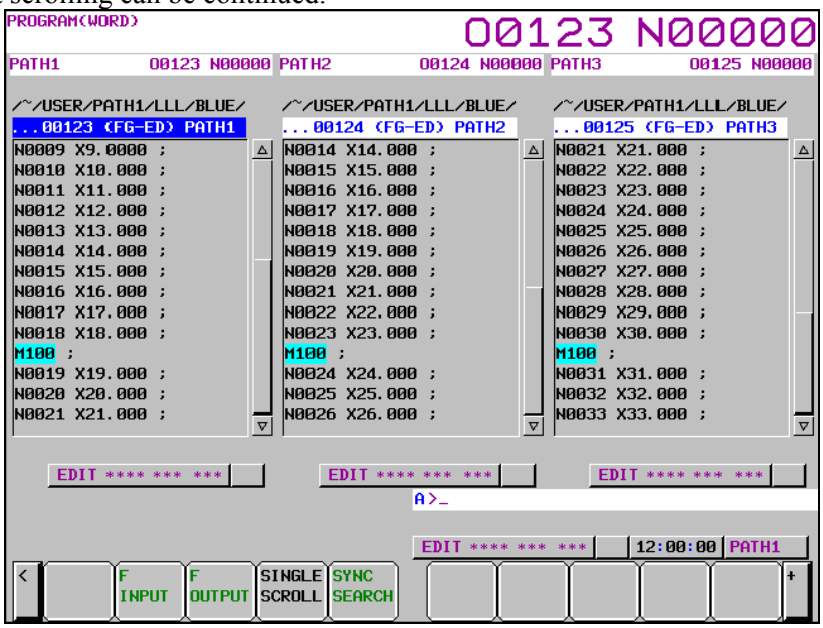


Fig. 10.15.2 (d) All programs displayed simultaneously are in the scroll waiting state

Release of scroll waiting

If, in the scroll waiting state, the cursor for the program for another path is moved to the beginning or end of the program, a confirmation message and soft keys are displayed as shown in Fig. 10.15.2 (e).
To release the scroll waiting state, press the soft key [EXEC].
To cancel the release of the scroll waiting state, press the soft key [CAN].

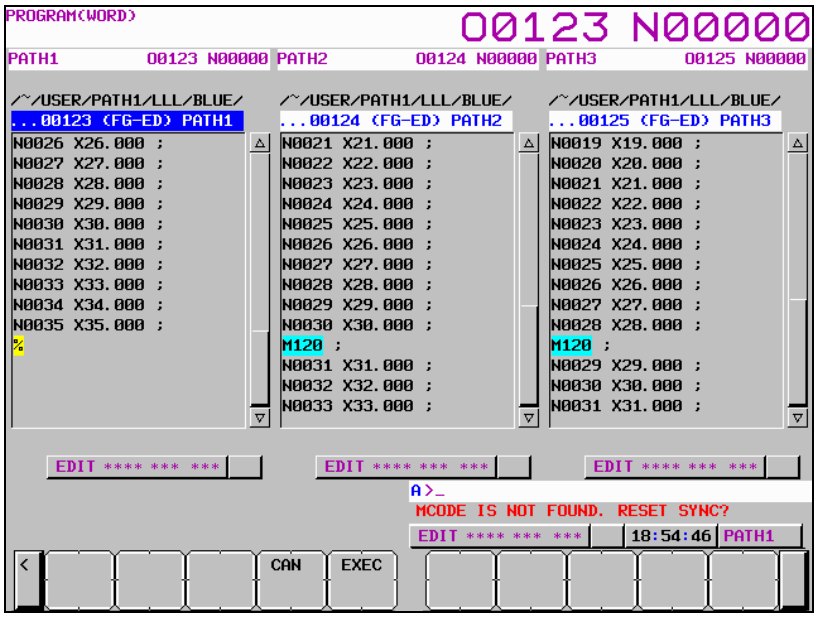


Fig. 10.15.2 (e) Display of the confirmation message for the release of the scroll waiting state

Waiting M-code search

By doing a waiting M-code search, it is possible to simultaneously move the cursors to the blocks that contain specified waiting-M code in all programs being edited simultaneously.

To do a waiting M-code search, press the soft key [SYNC SEARCH] in the simultaneous scroll mode. The soft keys shown in Fig. 10.15.2 (f) are displayed.

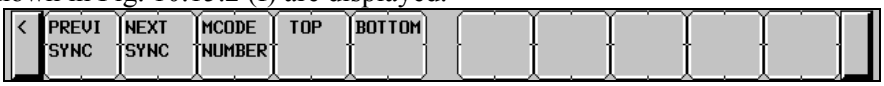


Fig. 10.15.2 (f) Soft keys for doing a waiting M-code search

[PREVI SYNC] Searches for a waiting M-code in the up direction, starting at the cursor position in the program to be edited.
The cursors of the paths specified for waiting move to the same waiting M-code.

[NEXT SYNC] Searches for a waiting M-code in the down direction, starting at the cursor position in the program to be edited.
The cursors of the paths specified for waiting move to the same waiting M-code.

[MCODE NUMBER] Searches for a specified waiting M-code. Only an M-code or an M-code number can be specified. The cursors move to the specified waiting M-code in all programs being edited simultaneously.

[TOP] In all programs being edited simultaneously, moves the cursors to the beginning of the programs.

[BOTTOM] In all programs being edited simultaneously, moves the cursors to the end of the programs.

If, as a result of a waiting M-code search, no waiting M-code is found, the cursor moves to the end of the program. In this case, the warning, "SYNC POINT IS NOT FOUND.", is displayed.

Restrictions

The simultaneous scroll function cannot be used in character editing or in background editing.

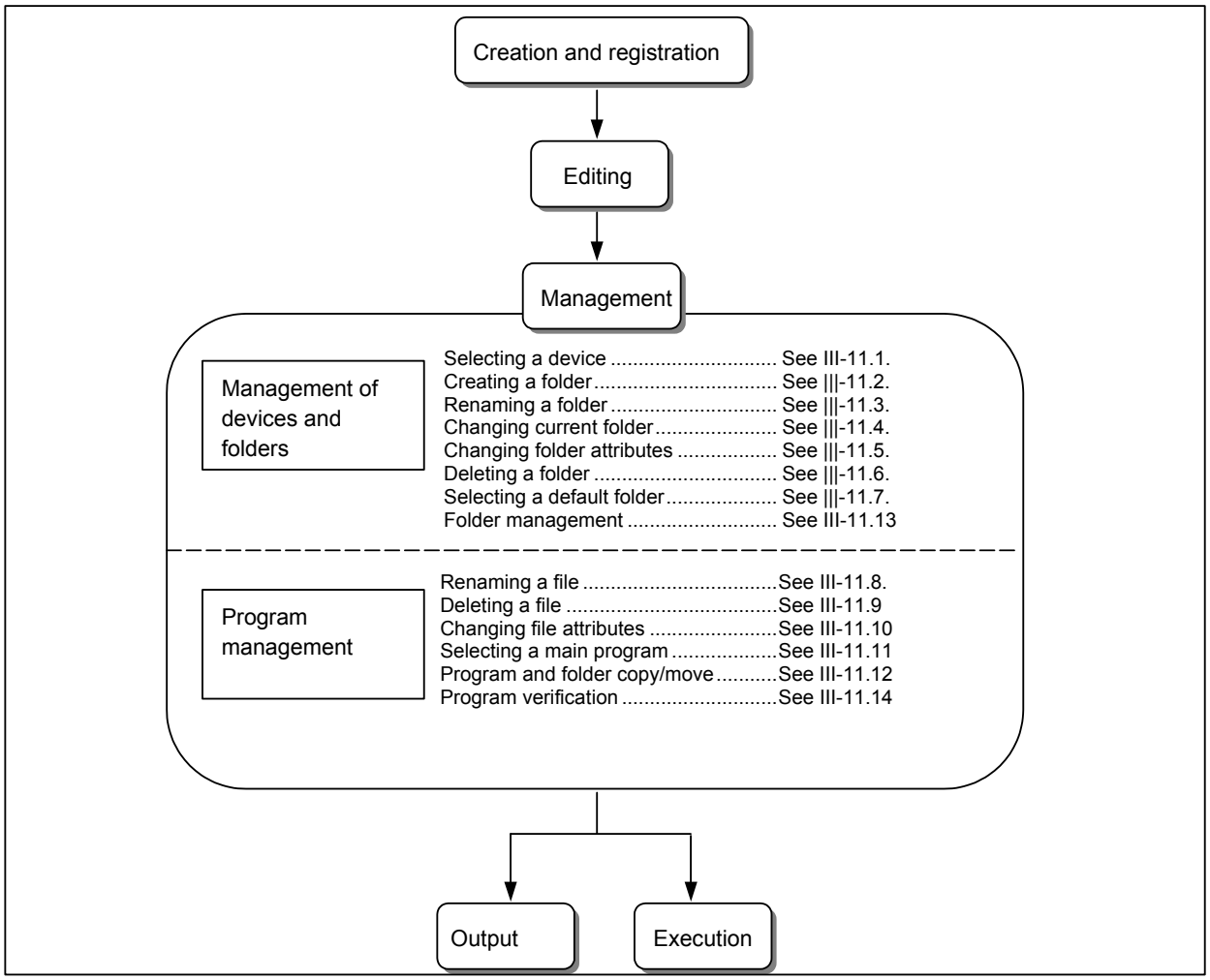
11 PROGRAM MANAGEMENT

Program management functions are classified into the following two types:

- Functions for folders
- Functions for programs

Functions for folders include creation, deletion, change of names and attributes, and so on.
Functions for programs include main program selection, deletion, change of names and attributes, and so on.

About program management, refer to "PROGRAM MANAGEMENT" in PROGRAMMING additionally.



11.1 SELECTING A DEVICE

When the fast data server function (option) is provided, a program storage device can be selected. This section explains the selection procedure.

Procedure for selecting a device

1 Press the function key .
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [DEVICECHANGE].
5 Press the soft key for the desired device.
Devices that can be selected are listed below (Table11.1 (a)):

Table11.1 (a)

Soft key	Device name
CNC MEM	CNC part program storage memory
MEM CARD	Program storage file (named "FANUCPRG.BIN")
EMB ETHER	Embedded Ethernet
DTSVR	Data Server
DTSVR HOST	Data Server Host
MEMORY CARD	Memory card
FLOPPY	Floppy cassette (Handy File)
USB MEM	USB memory

11.1.1 Selecting a Memory Card Program as a Device

Overview

By selecting a memory card including a program storage file (named "FANUCPRG.BIN") as a device, memory operation can be performed with the program in the program storage file selected as the main program.
In addition, the content of a program storage file can be displayed on the program list screen or a program in a program storage file can be edited on the program edit screen.
A program storage file can be created using a memory card program tool (A08B-9010-J700#ZZ11) on a commercially available personal computer. To be used, the program storage file must be written to a memory card formatted in FAT16 format.
(A program held in a program storage file is hereinafter referred to as a memory card program. Moreover, a memory card storing a program storage file is referred to as a program storage memory card.)

NOTE
Any program in a USB memory cannot be selected or edited.

Procedure for selecting a device

1 Press the function key .
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [DEVICECHANGE].
5 Press the soft key [MEMCARD].

NOTE
1 A FAT16-formatted memory card containing the program storage file FANUCPRG.BIN is recognized as a program storage memory card.
2 For a program storage memory card containing more than 63 folders and programs, the option for extending the number of memory card program registrations is required. The option for extending the number of memory card program registrations is applicable to a program storage memory card containing a maximum of 1000 folders and programs.

Procedure for removing a device

When a program storage memory card is replaced or a memory card is used for normal usage such as data input/output, clear the recognition of the program storage memory card with removal operation.

1 Press the function key 

.
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [DEVICECHANGE].
5 Press the soft key [DETACH].

NOTE
1 This soft key appears when the CNC recognizes a program storage memory card during device change operation.
2 This operation is enabled only in EDIT mode or MEM mode.
When a memory card program is selected in the main programs of multiple paths in a multi-path control system, set the modes of all paths to EDIT mode or MEM mode.
3 When the default folder is a folder in a program storage file, it is changed to "//CNC_MEM/" by a removal operation.
4 When the main program is a memory card program, the main program enters the unselected state by a removal operation.

Explanation

- About operation

A memory card program can be selected as a main program to perform memory operation.
Memory operation has the following features:

- Subprogram call nesting is allowed.
- Macro program call nesting is allowed.
- In a custom macro, a control command using a GOTO statement/WHILE statement can be specified.
- With the T series, a multiple repetitive canned cycle for turning can be specified.

NOTE
To use the functions for calling macros, using custom macros, and using a multiple repetitive canned cycle for turning, the options corresponding to the respective functions are required.

- Selection as a main program

As a main program to be automatically executed in the MEM mode, a memory card program can be selected.

- **Sub program (call using M98/G72.1/G72.2)**
- **Macro program (call using G65/G66/G66.1/M96)**

The following subprogram/macro program held in the same folder as containing the main program is called:

- Sub program call (M98)
- Macro call (Simple macro call G65 / Modal call G66, G66.1)
- Macro interrupt (M96)
- Figure copying (G72.1, G72.2)

If the program cannot be found in the same folder as the main program, the following folder is searched:

- Common program folder on the CNC_MEM device (CNC program storage memory)

NOTE
To use the functions for macro call, macro interrupt, and figure copying, the options corresponding to the respective functions are required.

- **Sub program (call using M code/S code/T code/particular address/the second auxiliary function)**
- **Macro program (call using G code/M code)**

The following subprogram/macro program calls a program from the CNC_MEM device (CNC program storage memory):

- Subprogram call using M code/S code/T code/particular addresses/the second auxiliary function
- Macro call using G code/M code

The folders set as search targets are searched in the following order, and the first found program is called:

1 Common program folder among the initial folders
2 MTB-dedicated folder 2 among the initial folders
3 MTB-dedicated folder 1 among the initial folders
4 System folder among the initial folders

Search target folders are set in parameter No. 3457.

NOTE
For a memory card program, subprogram call using M code/S code/T code/particular addresses/the second auxiliary function or macro call using G code/M code can be specified. However, a program on the CNC_MEM device (CNC program storage memory) is called.

- **External program number search / External workpiece number search**

A program on a program storage memory card can be searched for with the external program number search function or external workpiece number search function.

Limitation

For a memory card program, M198 cannot be specified. Moreover, no memory card program can be called from a program on the CNC_MEM device (CNC program storage memory) by specifying M198.

When a setting is made to enable an external device subprogram call from a memory card (M198) or DNC operation from a memory card (bit 7 (MNC) of parameter No. 0138 = 1), the content of program storage file cannot be displayed during automatic operation.

When a program storage memory card is selected, the memory card cannot be used for the ordinary purposes listed below. To use a memory card in such a case, perform a "removal" operation to cancel the recognition of the program storage memory card.

- ALL I/O screen
  Display of the contents of a memory card, and input/output data to and from a memory card
- PMC data I/O screen
  Display of the contents of a memory card, and input/output to and from a memory card
- Program directory screen
  Input/output program data to and from a memory card
- External device subprogram call (M198) operation
  Subprogram call (M198) with a memory card set as an external device
- DNC operation
  DNC operation from a memory card

⚠ CAUTION

1 Do not remove the memory card when a program that specifies a write to the memory card is being edited. The data can be destructed.
2 If an editing operation is completed, the results of editing are preserved even when the power to the CNC is turned off.
3 When removing the memory card, be sure to perform a "removal" operation. If the memory card is removed without performing a "removal" operation and an attempt is made to access the memory card, the alarm (SR1964) or alarm (IO1030) is issued.
  If the card is removed inadvertently, insert the card again and perform a "removal" operation.
  When an alarm is issued, perform the following operation:
  - When the alarm (SR1964) is issued
    Reset the alarm after performing a "removal" operation.
  - When the alarm (IO1030) is issued
    The alarm can be reset only by turning off the power to the CNC.
4 There are cases in which when a memory card is replaced with another, the CNC cannot detect the replacement. Thus, it is risky to replace a memory card without performing a "removal" operation, and this should never be attempted.

- Operation of creation, edition, and management of a program

When “memory card program as a device” is selected, operation of creation, edition, and management of a program is below (Table 11.1.1 (b)):

Table 11.1.1 (b)

Item	Usable
Creation of a program	Unusable
Edition prohibition attribute	Unusable
Inserting, alteration, and deletion a Word	Usable
Deletion of a block	Usable
Program search	Usable
Sequence number search	Usable
Deletion of a program	Unusable
Editing a custom macro	Usable
Password function	Unusable
Editing a program character	Usable
Program and folder copy/move function	Usable
Key and program encryption	Unusable
Selecting a device	Usable
Creation of a folder	Unusable
Renaming of a folder	Unusable
Changing the attribute of a folder	Unusable

Item	Usable
Deletion of a folder	Unusable
Selecting a default folder	Usable
Renaming of a file	Unusable
Deletion of a file	Unusable
Changing the attribute of a file	Unusable
Selecting a main program	Usable
Input/output of program	Unusable

NOTE
In memory card program device, programs are copied from memory card program device to CNC built-in memory device or to data server device, or copied from CNC built-in memory device or from data server device to memory card program device.

11.1.2 Selecting a From Cassette as a Device

The Handy File can be connected to the NC to display a list of programs in a floppy cassette.

Procedure for selecting a device

1 Set the relevant parameters so that communication with the Handy File is enabled.
2 Press the function key 

.
3 Press the soft key [FOLDER].
4 Press the soft key [(OPRT)].
5 Press the soft key [DEVICECHANGE].
6 Press the soft key [FLOPPY].

Limitation

- Floppy disk directory display during schedule operation

During schedule operation, directories in a floppy disk cannot be displayed in a background edit.

- Operation of creation, edition, and management of a program

When “floppy cassette as a device” is selected, operation of creation, edition, and management of a program is below (Table 11.1.2 (a)):

Table 11.1.2 (a)

Item	Usable
Creation of a program	Unusable
Edition prohibition attribute	Unusable
Inserting, alteration, and deletion a Word	Unusable
Deletion of a block	Unusable
Program search	Usable
Sequence number search	Unusable
Deletion of a program	Usable
Editing a custom macro	Unusable
Password function	Unusable
Editing a program character	Unusable
Program and folder copy/move function	Unusable
Key and program encryption	Unusable
Selecting a device	Usable
Creation of a folder	Unusable
Renaming of a folder	Unusable

Item	Usable
Changing the attribute of a folder	Unusable
Deletion of a folder	Unusable
Selecting a default folder	Unusable
Renaming of a file	Unusable
Deletion of a file	Usable
Changing the attribute of a file	Unusable
Selecting a main program	Unusable
Input/output of program	Usable

11.1.3 Selecting a USB Memory as a Device

When a commercially available USB memory is inserted into the USB memory interface of the CNC, the CNC recognizes the USB memory as an external input/output device.

The following operations can be performed for a USB memory:
- Listing the files and folders in the USB memory
- Renaming and creating a file or folder in the USB memory
- Inputting and outputting an NC program between the CNC part program storage memory and USB memory

Procedure for selecting a device

1 Press the function key 

.
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [DEVICECHANGE].
5 Press the soft key [USB MEM].

Limitation

There are the following restrictions on the USB function.
(1) Only one USB memory can be recognized. (No USB hub can be recognized.)
(2) Up to 32 characters can be used for a file or folder name.
If a file or folder name is longer than 32 characters, the 32nd character is replaced with a tilde (~) and the subsequent characters are omitted when displayed. All operations including management operation and input/output cannot be performed for a file or folder whose name is omitted with a tilde (~).
(3) The maximum number of folder levels is 6 (excluding the root folder).
(4) The maximum number of files and folders in a folder is 512.
(5) If a non-ASCII character (alphanumeric character) is used in a file or folder name, it cannot be displayed correctly.

For detailed specifications of and restrictions on a USB memory, see the section related to the USB function.

NOTE
Remove a USB memory only when “REMOVE OK” is displayed as the USB memory status, or data may be damaged.

- Operation of creation, edition, and management of a program

When “USB memory as a device” is selected, operation of creation, edition, and management of a program is below (Table 11.1.3 (a)):

Table 11.1.3 (a)

Item	Usable
Creation of a program	Unusable
Edition prohibition attribute	Unusable
Inserting, alteration, and deletion a Word	Unusable
Deletion of a block	Unusable
Program search	Usable
Sequence number search	Unusable
Deletion of a program	Usable
Editing a custom macro	Unusable
Password function	Unusable
Editing a program character	Unusable
Program and folder copy/move function	Usable
Key and program encryption	Unusable
Selecting a device	Usable
Creation of a folder	Usable
Renaming of a folder	Usable
Changing the attribute of a folder	Unusable
Deletion of a folder	Usable
Selecting a default folder	Unusable
Renaming of a file	Usable
Deletion of a file	Usable
Changing the attribute of a file	Unusable
Selecting a main program	Unusable
Input/output of program	Usable

NOTE
In USB device, programs are copied from USB device to CNC built-in memory device or from CNC built-in memory device to USB device.

11.2 CREATING A FOLDER

This section explains the procedure for creating a folder.

Procedure for creating a folder

1 Select EDIT mode.
2 Press the function key 

.
3 Move to the folder in which you want to create a folder.
Use the cursor keys  and  to move among folders. After selecting the folder, press the 

key.
4 Press the soft key [(OPRT)].
5 Type folder name.
6 Press the soft key [CREATE FOLDER].

NOTE
1 Each folder name must be unique within the same folder.
2 Each time a folder is created, the number of programs that can be registered decreases by one.

NOTE
3 Depending on the operation status and protection status, a folder cannot sometimes be created.

11.3 RENAMING A FOLDER

This section explains the procedure for renaming a folder.

Procedure for renaming a folder

1 Select EDIT mode.
2 Press the function key .
3 Press the soft key [FOLDER].
4 Select the folder that you want to rename.
To select a folder, use the cursor keys  and .
5 Press the soft key [(OPRT)].
6 Type new folder name.
7 Press the soft key [RENAME].

NOTE
1 You cannot rename the initial folders.
2 Each folder name must be unique within the same folder.
3 Depending on the operation status and protection status, a folder cannot sometimes be renamed.

11.4 CHANGING CURRENT FOLDER

Current folder is selected folder in program contents display and program folder screen.

Procedure for changing current folder in folder screen

Changing to the upper folder

1 Press the function key .
2 Press the soft key [FOLDER].
3 Move the cursor to “RETURN TO UPPER FOLDER”.
4 Press the MDI key .

Changing to the under folder

1 Press the function key .
2 Press the soft key [FOLDER].
3 Move the cursor to the folder to select.
4 Press the MDI key .

Procedure for changing current folder in folder tree display.

In folder tree display, current folder is changed by moving cursor on the tree.

Changing to the upper folder

1 Press the function key PROG.
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [TREE LIST].
5 Press the cursor key ←.

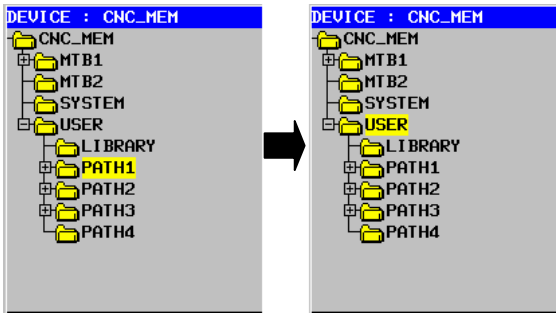


Fig.11.4 (a)

Changing to the under folder

1 Press the function key PROG.
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [TREE LIST].
5 Press the cursor key →.

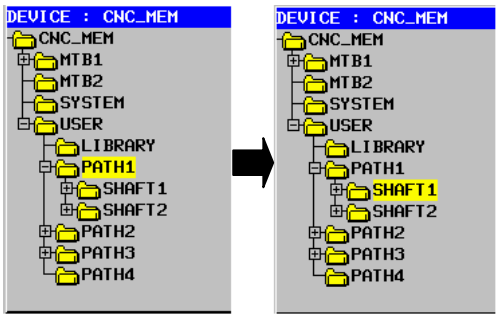


Fig.11.4 (b)

Changing to the next folder

1 Press the function key PROG.
2 Press the soft key [FOLDER].
3 Press the soft key [(OPRT)].
4 Press the soft key [TREE LSIT].
5 Press the cursor key ↑ or ↓.

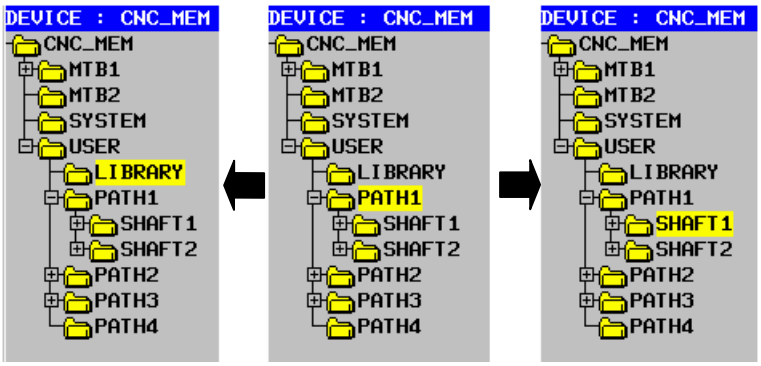


Fig.11.4 (c)

11.5 CHANGING FOLDER ATTRIBUTES

This section explains the procedure for changing the attribute of a folder (edit disable or edit/display disable).

Procedure for changing folder attributes

1 Select EDIT mode.
2 Press the function key 

.
3 Press the soft key [FOLDER].
4 Select the folder of which attribute is to be changed.
To select a folder, use the cursor keys  and .
5 Press the soft key [(OPRT)].
6 Press the soft key [DETAILON].
7 Press the soft key [CHANGE ATTR].
- To disable editing, press the soft key [EDIT DISABL].
- To enable editing, press the soft key [EDIT ENABLE].
- To disable editing and display, press the soft key [DISP DISABL].
- To enable editing and display, press the soft key [DISP ENABLE].

NOTE
1 Depending on the operation status and protection status, the attribute of a folder cannot sometimes be changed.
2 When the edit disable attribute is set for a folder, editing of folders and files in that folder is disabled.
3 When the edit/display disable attribute is set for a folder, editing and display of folders and files in that folder is disabled, and these folders and files are not displayed.
4 The items that can be set vary depending on the status of parameters and so on.

11.6 DELETING A FOLDER

This section explains the procedure for deleting a folder.

Procedure for deleting a folder

1 Select EDIT mode.

2 Press the function key .
3 Press the soft key [FOLDER].
4 Select the folder that you want to delete.
To select a folder, use the cursor keys  and .
5 Press the soft key [(OPRT)].
6 Press the soft key [DELETE].
- To perform the deletion, press the soft key [EXEC].
- To cancel the deletion, press the soft key [CANCEL].

NOTE
1 You cannot delete the initial folders.
2 A folder cannot be deleted unless the folder is empty.
(An empty folder means that the folder contains neither folders nor files.)
3 If a folder contains a folder or file having the edit/display disable attribute, the folder may seem to be empty when displayed, but the folder is not actually empty, so it cannot be deleted.
4 Depending on the operation status and protection status, a folder cannot sometimes be deleted.

11.7 SELECTING A DEFAULT FOLDER

This section explains the procedure for selecting a foreground or background default folder.
What is a default folder is described in "PROGRAM MANAGEMENT" in PROGGRAMMING.

Procedure for selecting a default folder

1 Select EDIT mode.
2 Press the function key .
3 Press the soft key [FOLDER].
4 Move to the folder that you want to select as the default folder.
Use the cursor keys  and  to move among folders. After selecting the folder, press the  key.
5 Press the soft key [(OPRT)].
6 - To select the foreground, press the soft key [FORE CHANGE].
- To select the background, press the soft key [BACK CHANGE].

NOTE
1 When the foreground or background default folder is not set, the path folder, which is an initial folder, is assumed.
2 The settings for the foreground and background default folders are stored in the default folder setting file.
3 When a program file, program folder, or program folder management file is cleared, the default folder setting file is also cleared at the same time.

11.8 RENAMING A FILE

This section explains the procedure for renaming a file.

Procedure for renaming a file

1 Select EDIT mode.
2 Press the function key .
3 Press the soft key [FOLDER].
4 Move to the folder containing the file that you want to rename.
Use the cursor keys  and  to move among folders. After selecting the folder, press the  key.
5 Select the file that you want to rename.
To select a file, use the cursor keys  and .
6 Press the soft key [(OPRT)].
7 Type new file name.
8 Press the soft key [RENAME].

NOTE
1 Each file name must be unique within the same file.
2 When the assigned file name cannot be treated as a program number, the program in the file is restricted as follows:
- Specification by program number is impossible. (Such as a subprogram call)
- Information output by program number is impossible.

3 Depending on the operation status and protection status, a file cannot sometimes be renamed.

11.9 DELETING A FILE

This section explains the procedure for deleting a file.

Procedure for deleting a file

1 Select EDIT mode.
2 Press the function key .
3 Press the soft key [FOLDER].
4 Move to the folder containing the file that you want to delete.
Use the cursor keys  and  to move among folders. After selecting the folder, press the  key.
5 Select the file that you want to delete.
To select a file, use the cursor keys  and .
6 Press the soft key [(OPRT)].
7 Press the soft key [DELETE].
- To perform the deletion, press the soft key [EXEC].

- To cancel the deletion, press the soft key [CANCEL].

NOTE
Depending on the operation status and protection status, a file cannot sometimes be deleted.

11.10 CHANGING FILE ATTRIBUTES

This section explains the procedure for changing the attribute of a file (edit disable, edit/display disable, encoding, or protection of data at eight levels).

Procedure for selecting the attribute of a file

1 Select EDIT mode.
2 Press the function key 

.
3 Press the soft key [FOLDER].
4 Move to the folder containing the file of which attribute is to be changed.
Use the cursor keys  and  to move among folders. After selecting the folder, press the 

key.
5 Select the file of which attribute is to be changed.
To select a file, use the cursor keys  and .
6 Press the soft key [(OPRT)].
7 Press the soft key [DETAILON].
8 Press the continuous menu key  until the desired soft key appears.
9
- To disable editing, press the soft key [EDIT DISABL].
- To enable editing, press the soft key [EDIT ENABLE].
- To disable editing and display, press the soft key [DISP DISABL].
- To enable editing/display, press the soft key [DISP ENABLE].
- To set encoding, press the soft key [ENCODESET].
- To cancel encoding, press the soft key [ENCODE RESET].
- To change the change protection level, type a change protection level, then press the soft key [CHANGELEVEL].
- To change the output protection level, type an output protection level, then press soft key [OUT LEVEL].

NOTE
1 Depending on the operation status and protection status, a file cannot sometimes be deleted.
2 The items that can be set vary depending on the status of options, parameters, and so on.
3 To change the display disable attribute, release protection of programs in advance. Set the same value as the password (parameter No. 3210 (PSW)) in parameter No. 3211 (KEY) to release the protection.

11.11 SELECTING A MAIN PROGRAM

This section explains the procedure for selecting a main program.

Procedure for selecting a main program

1 Select EDIT mode.
2 Press the function key PROG.
3 Press the soft key [FOLDER].
4 Move to the folder containing the file that you want to use as a main program.
Use the cursor keys ↓ and ↑ to move among folders. After selecting the folder, press the INPUT key.
5 Select the file that you want to use as the main program.
To select a file, use the cursor keys ↓ and ↑.
6 Press the soft key [(OPRT)].
7 Press the soft key [MAIN PROGRM].

NOTE
Depending on the operation status and protection status, the main program cannot sometimes be selected.

11.12 PROGRAM AND FOLDER COPY/MOVE FUNCTION

Overview

Programs and folders can be copied or moved from folder to folder.

Procedure for copying a program or a folder

1. Press the function key PROG.
2. Press the chapter selection soft key [FOLDER].
The program folder screen in Fig.11.12 (a) appears:

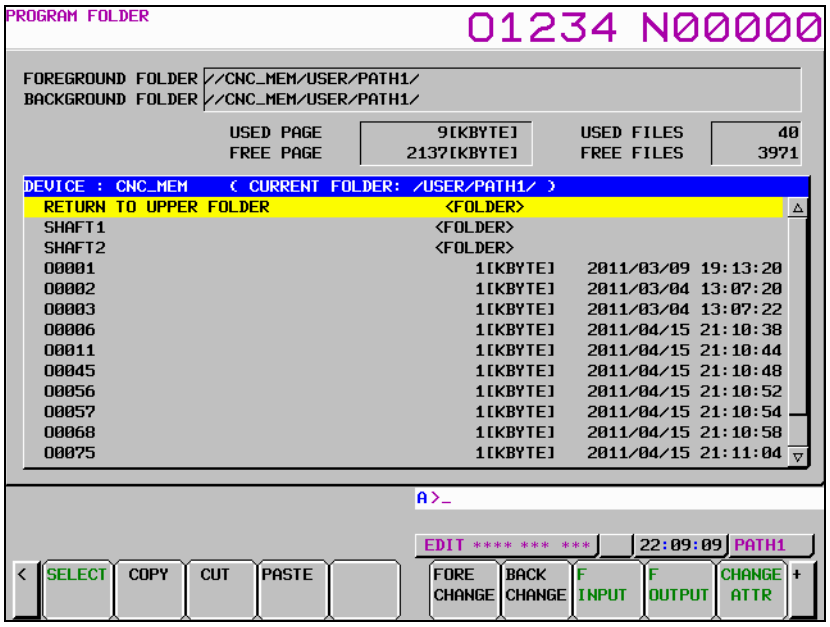


Fig.11.12 (a) Program folder screen

3. Display the folder from which you want to copy or move the program or the folder. Move the cursor to the folder on the screen and press the key [INPUT] to move to the folder. Move the cursor to "RETURN TO UPPER FOLDER" on the screen and press the key [INPUT] to move the upper folder.
4. Press the soft key [(OPRT)].
5. Press the continuous menu key [▷] until the soft key [COPY] appears.
6. Move the cursor to the program or the folder that you want to select.
7. Press the soft key [COPY]. The program or folder is selected, and the background color becomes selected color (default: blue). If you want to move the program or the folder, press the soft key [CUT].
8. Display the folder to which you want to copy or move the program or folder.
9. Press the soft key [PASTE]. If you want to rename the program or folder, type new name before pressing [PASTE].

Procedure for copying some programs and folders

Selection by range

1. Press the function key [PROG].
2. Press the chapter selection soft key [FOLDER].
3. Display the folder from which you want to copy or move the program or the folder. Move the cursor to the folder on the screen and press the key [INPUT] to move to the folder. Move the cursor to "RETURN TO UPPER FOLDER" on the screen and press the key [INPUT] to move the upper folder.
4. Press the soft key [(OPRT)].
5. Press the continuous menu key [▷] until the soft key [SELECT] appears.
6. Press the soft key [SELECT].
7. Press the soft key [RANGE SELECT]. In addition, move the cursor up and down to select the programs and the folders. The background color of selected programs and folders becomes selected color (default: blue).
8. If the selected range is determined, press the soft key [COPY]. If you want to move the program or folder, press the soft key [CUT].
9. Change the current folder to the destination folder.

10. Press the soft key [PASTE].

Selection by each program or folder

1. Press the function key .
2. Press the chapter selection soft key [FOLDER].
3. Press the soft key [(OPRT)].
4. Press the continuous menu key  until the soft key [COPY] appears.
5. Press the soft key [SELECT].
6. Move the cursor to program or folder to be copied.
7. Press the soft key [SELECT].
8. Repeat 6th and 7th operation until select is ended.
9. When selected programs or folders are determined, press the soft key [COPY]. If you want to move the program or folder, press the soft key [CUT].
10. Change the current folder to the destination folder.
11. Press the soft key [PASTE].

NOTE
Programs or folders up to 10 can be selected.

About same name programs and folders

When the same name of program or folder exists at the move destinations, confirmation message "OVERWRITE? (program or folder name)" is displayed.

- When the soft key [YES] is pressed, the program/folder is overwritten. If the same name programs/folders appear again, the confirmation message is displayed.
- When the soft key [NO] is pressed, copying or moving operation of the program/folder is skipped. If the same name programs/folders appear again, the confirmation message is displayed.
- When the soft key [YES ALL] is pressed, the program/folder is overwritten. Even if the same name programs/folders appear again, they are also overwritten.
- When soft key [NO ALL] is pressed, copying or moving operation of program/folder is skipped. If the same name programs/folders appear again, they are also skipped.
- When the soft key [CANCEL] or [<] is pressed, copying or moving operation is canceled.

During the overwrite confirmation is displayed and the screen is changed, the move operation is canceled.

Restrictions

If the following conditions are corresponded, copy and move operation is invalid.

- The program data protection is enable.(The memory protect signal KEY3 is OFF)
- The top of the origin folder of copy/move or destination folder of copy/move has the edit disable attribute.
- When the copy and move operation is performed to sub-folder of the copy/move origin folder.

Programs and folders, which correspond to the following conditions, cannot be copied and moved.

- The copy/move origin or destinations programs/folders are protected by the edit disable attribute.
- The copy/move origin or destinations programs/folders are protected by 8 level data protection function.
- The copy/Move destinations program is selected as a main program.
- The depth of the path is exceeded 6 classes after the folders are copied/moved.

11.12.1 Copy and movement between different devices

Overview

It is enable to copy and move the files between different devices.
The files can be copied/moved in a same way.

Restrictions

About file operation restrictions on each device

A copy or movement between the following devices is possible.

Table11.12.1 (a) Copy and movement between different devices

		Destination			
		CNC built-in memory	**Data server**	**Memory card**	**USB**
Origin	**CNC built-in memory**	A	A	B	B
	Data server	A	A	C	C
	Memory card program	A	A	C	C
	Memory card	B	C	C	C
	USB	B	C	C	C

A : Programs and folders can be copied and moved
B : Programs can be copied and moved. Folders cannot be copied and moved.
(NOTICE: Memory card program device can be only copied.)
C : Programs and folders cannot be copied and moved.

NOTE
1 Memory card program device has following restrictions.
(i) Programs cannot be moved in memory card program device.
(ii) Memory card program device cannot be used with memory card device.
2 Memory card device has following restrictions.
(i) Memory card device cannot be used with memory card program device.
(ii) When the programs are copied, place all paths in the EDIT mode.
(iii) When the programs are copied from CNC built-in memory device to memory card device, the behavior follows the setting of bit 1 (COW) of parameter No.11308.
(iv) When the programs are copied from memory card device to CNC built-in memory device, the behavior follows the setting of bit 2 (REP) of parameter No.3201.
3 USB device has following restrictions.
(i) When the programs are copied, place all paths in the EDIT mode.

Copy of file including some programs

When the file including some program is copied, don’t input the file name to the key input buffer.
When some programs are included in the file of the copy origin, if the character string is input to the key input buffer, only the program of the head in the file is read to CNC MEM with program name input to the key input buffer.

File name on memory card

When the program with 8 character or more name of the program in the program memory on CNC is output, it is output on the memory card as follows.

For eight digits O numbers ('O'+ numeric parts)

- The numeric parts are 8 digits or more :
  The file name is changed to eight numerical value digits except first 'O'.
  Example) O12345678 → 12345678
- The numeric parts are less than 8 digits :
  One digit is deleted '0' and program name is changed to seven digits of 'O' + numerical value.
  Example) O00012345 → O0012345

For arbitrary program name (9 characters or more)

- Program name is changed to the file name that rounds down since the ninth character.
  Example) 123456789 → 12345678

The file name output on the memory card might overlap by the above operation. In this case, it is output according to the setting of bit 1 (COW) of parameter No.11308.

11.13 FOLDER MANAGEMENT

This function restricts the folder operation in program management. As a result, operator can manage the program without the complex folder operation. Following two specifications can be selected by the parameter.

1) The folders under the path folders can be used for program management.
2) Only the path folders can be used for program management.

Path folders are the one of the initial folders. The folder exists under //CNC_MEM/USER, and the number is same to the system path. Please refer to USER'S MANUAL (B-64484EN, II 12 PROGRAM MANAGEMENT) for details.

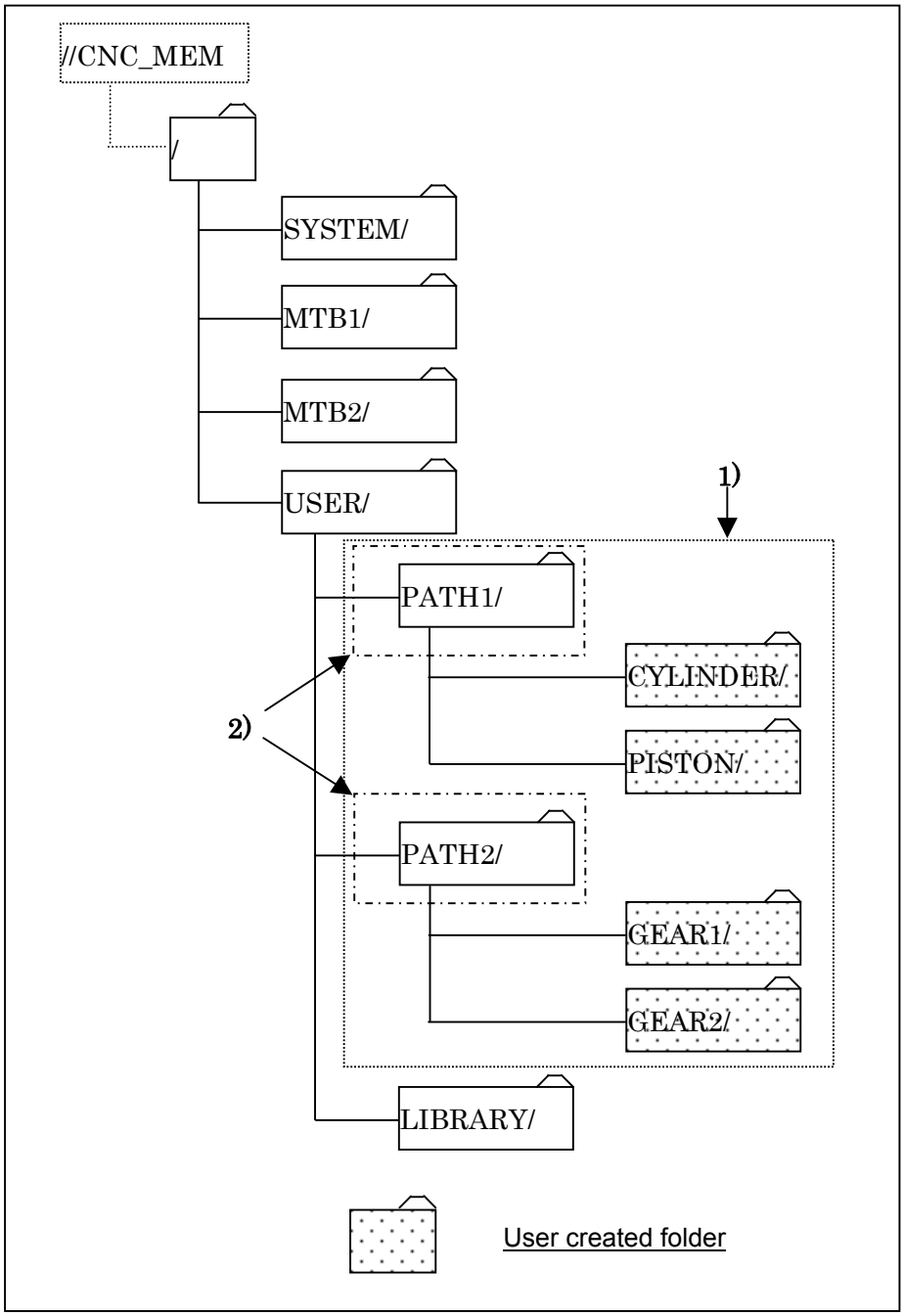


Fig. 11.13 (a) Folder that can be used by each setting

11.13.1 Program Management under the Path Folder

If parameter FPF (No.11302#6) is set to “1”, the operation on the program folder screen becomes as follows. At this time, it is necessary to set parameter CFP (No.11304#7) to “0”.

Table 11.13.1 (a) Restriction of operation

Operation		Working
Moving of the folder	Under the path folder	Possible
	Upper from the path folder	Impossible
Creating of user folder		Possible
Changing of foreground default folder		Possible
Changing of background default folder		Possible

Display on program folder screen

Only the folder under the path folder corresponding to selected path can be displayed.
Warning message “CAN’T MOVE TO FOLDER” is displayed and cannot move when the folder is operated to move upper from the path folder.

Setting of foreground/background default folder

Only the folder under the path folder corresponding to selected path can be set to foreground/background default folder.

11.13.2 Program Management only in the Path Folder

If parameter CFP (No.11304#7) is set to 1, the operation on the program folder screen becomes as follows. At this time, the setting of parameter FPF (No.11302#6) is invalid.

Table 11.13.2 (a) Restriction of operation

Operation		Working
Moving of the folder	Under the path folder	Impossible
	Upper from the path folder	Impossible
Creating of user folder		Impossible
Changing of foreground default folder		Impossible
Changing of background default folder		Impossible

Display on program folder screen

Only the path folder corresponding to selected path can be displayed. “RETURN TO UPPER FOLDER” is not displayed. (Refer to “1)” in Fig. 11.13.2 (a) and Fig. 11.13.2 (b).)
The subfolder that has already been made under the path folder before the parameter CFP(No.11304#7) is made effective can be displayed. But it is impossible to move to the subfolder.

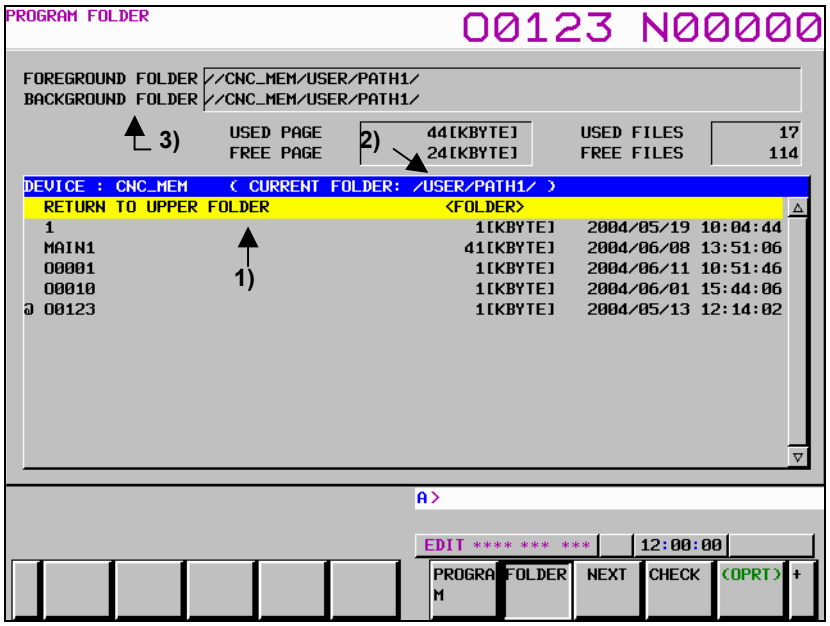


Fig. 11.13.2 (a) Program folder screen (parameter CFP(No.11304#7)=0)

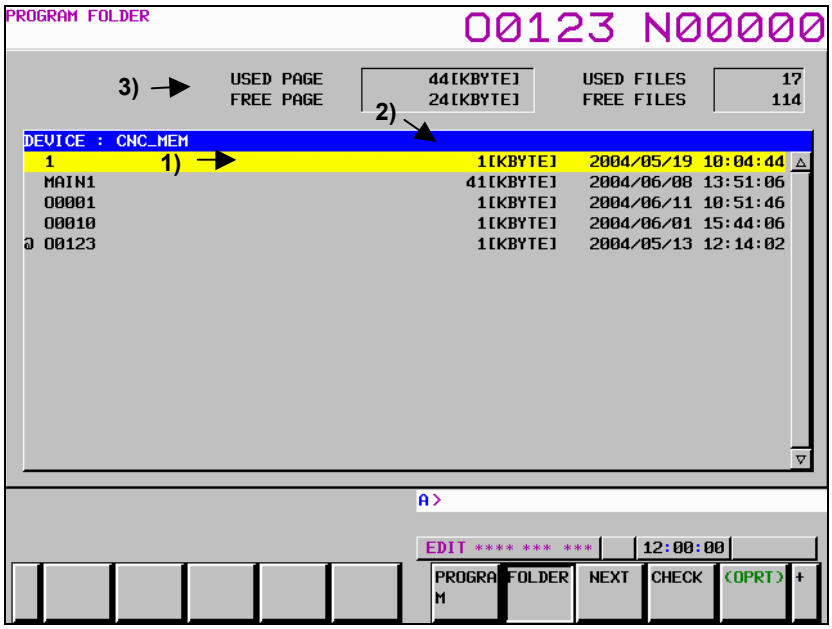


Fig. 11.13.2 (b) Program folder screen (parameter CFP(No.11304#7)=1)

Display of device name and current folder

The current folder name is not displayed though the device name is displayed so far. (Refer to "2)" in Fig. 11.13.2 (a) and Fig. 11.13.2 (b).)

Display and setting of foreground/background default folder

Foreground/background default folders are not displayed. (Refer to "3)" in Fig. 11.13.2 (a) and Fig. 11.13.2 (b).)

Moreover, foreground/background default folders cannot be set.

NOTE

Set foreground/background default folder to each path folder before the parameter CFP(No.11304#7) is made effective.

Soft key that is not displayed

The following soft key is not displayed, and cannot be operated.

- CREATE FOLDER 1)
- FORE CHANGE 2)
- BACK CHANGE 3)

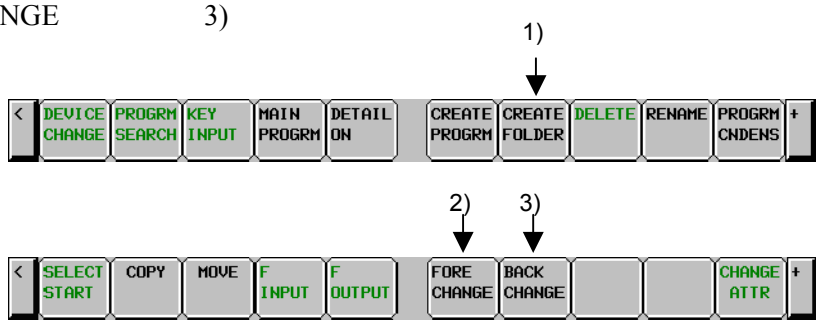


Fig11.13.2 (c) Soft key display for CNC_MEM device

11.13.3 Folder for Subprogram/Macro Calls

In the following subprogram call/macro call, the folder /USER/LIBRARY, /MTB12, /MTB2 and /SYSTEM are searched before the folder where there is the main program are searched.
There is a possibility being executed for the subprogram that the operator doesn't intend if the subprogram is registered to the above-mentioned folder even if the display and the operation are limited by this function.

-Subprogram call based on an M code
-Subprogram call based on a particular address
-Subprogram call based on a second auxiliary function code
-Macro call based on a S code
-Macro call based on a G code
-Macro call based on an M code
-Macro call based on a T code
-One-touch macro call

By setting 1 to the parameter SCC (No.3457#6), the folder where there is the main program is added to the top of search order.

1) Folder where the main program is stored. ← Added.
2) Common program folder of initial folder
3) MTB-dedicated folder 2 of initial folder
4) MTB-dedicated folder 1 of initial folder
5) System folder of initial folder

The folders of 2) though 5) can be excluded from search target folders by setting the parameters LIB(No.3457#0), MC2(No.3457#1), MC1(No.3457 #2), and SYS(No.3457#3).

11.14 PROGRAM VERIFICATION

This function enables to verify the differences of the programs in the I/O device and the programs in the CNC built-in memory.

Sequence of program verification

1. Make sure the I/O device is ready for input and output.
2. Press the EDIT switch on the machine operator's panel.
3. The program data protection is enabled on the machine operator's panel.(Turn off the memory protect signal KEY3)
4. Press the function key  and display the program list screen.
5. Press the soft key [(OPRT)].
6. Press the continuous menu key [+].
7. Press the soft key [F INPUT].
8. Set the file name in the I/O devices.
   Input the file name and press the soft key [F NAME].
   If the file name is not set, "ALL-PROG.TXT" is set.
9. Set the program name in the NC memory.
   Input program name and press the soft key [P SET].
   If the program name is not specified, the program names in the file which is set in above '8' are set.
10. Press the soft key [EXEC].
    Program verification is started and the "COMPARE" is blinked in lower right. When verification finishes, "COMPARE" disappears.
    If the program in the I/O device and the program in the CNC built-in memory are same, the message

"PROGRAM HAS BEEN VERIFIED" is displayed.
If they are different, the alarm (PS0079) "PROGRAM NOT MATCH" is issued

NOTE
If 8LV data protection function is valid, this function cannot be used.

Explanation

When one program is registered in a file

If one program is registered in the file, the program in the file and the program that is specified by [P SET] are verified.
If the program in the CNC built-in memory is not specified by [P SET], the program whose program name is same as program in the file are verified.

When some programs are registered in a file

Each program in the file which is specified by [F SET] and the program whose program name is same as the program name in the file are verified. When all verifications succeed, the message "PROGRAM HAS BEEN VERIFIED" is displayed.

NOTE
When the program is specified by [P SET], the alarm (PS0079) is issued as long as all of programs in the file are same as the program which is set by [P SET].

12 SETTING AND DISPLAYING DATA

To operate a CNC machine tool, various data must be set on the MDI unit for the CNC. The operator can monitor the state of operation with data displayed during operation.
This chapter describes how to display and set data for each function.

Chapter 12, "SETTING AND DISPLAYING DATA", consists of the following sections:

12.1 SCREENS DISPLAYED BY FUNCTION KEY 

.....1344
12.2 SCREENS DISPLAYED BY FUNCTION KEY 

.....1372
12.3 SCREENS DISPLAYED BY FUNCTION KEY 

.....1457
12.4 SCREENS DISPLAYED BY FUNCTION KEY 

.....1627
12.5 SCREENS DISPLAYED BY FUNCTION KEY 

.....1715
12.6 SWITCHING BETWEEN MULTI-PATH DISPLAY AND SINGLE-PATH DISPLAY FUNCTION 1718
12.7 FIVE AXES DISPLAY IN ONE SCREEN FOR THE 8.4-INCH DISPLAY UNIT 1722
12.8 PATH NAME EXPANSION DISPLAY FUNCTION 1725
12.9 SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION 1727
12.10 LOAD METER SCREEN 1729
12.11 DISPLAYING THE PROGRAM NUMBER/NAME, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION 1734

Explanation

- Screen transition chart

The screen transition for when each function key on the MDI unit is pressed is shown below. The subsections referenced for each screen are also shown. See the appropriate subsection for details of each screen and the setting procedure on the screen. See other chapters for screens not described in this chapter.

See Chapter 7 for the screen that appears when function key 

is pressed. In general, function key 

or 

is prepared by the machine tool builder and used for macros. Refer to the manual issued by the machine tool builder for the screen that appears when function key 

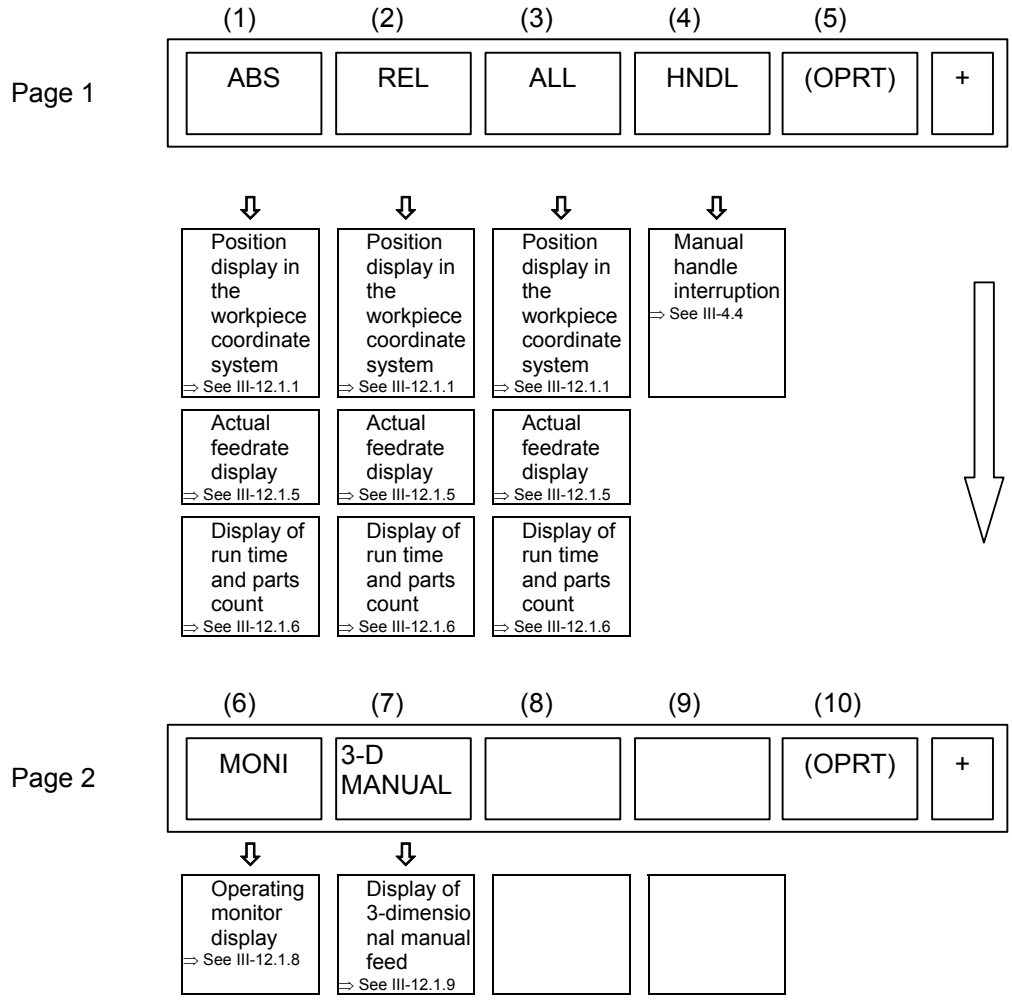
or 

is pressed.

- Data protection key

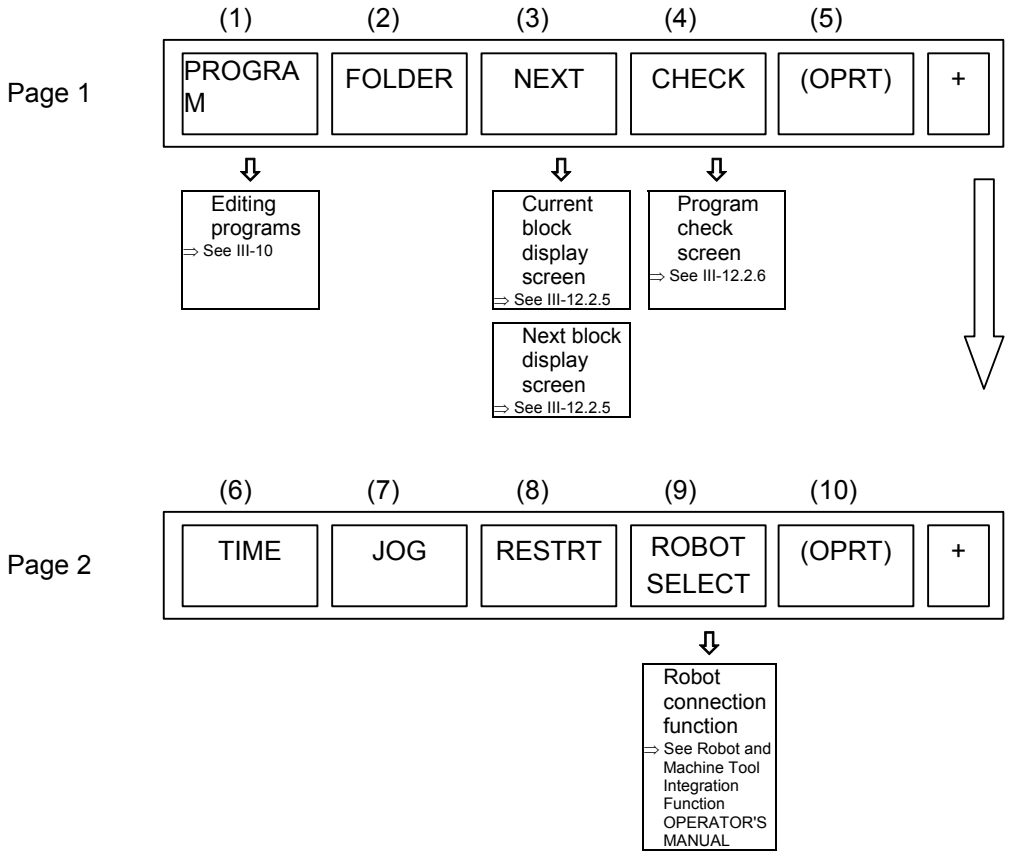
The machine may have a data protection key to protect part programs, tool compensation values, setting data, and custom macro variables. Refer to the manual issued by the machine tool builder for where the data protection key is located and how to use it.

Screen displayed when the function key POS is pressed (for 8.4/10.4-inch display unit)

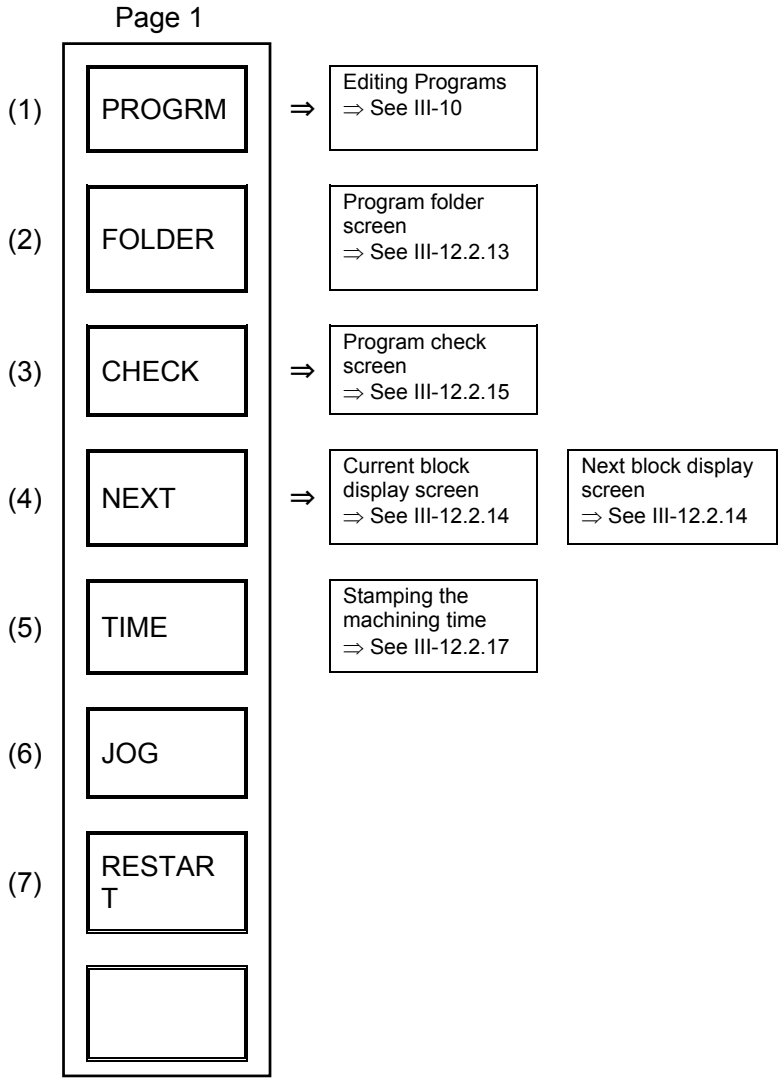


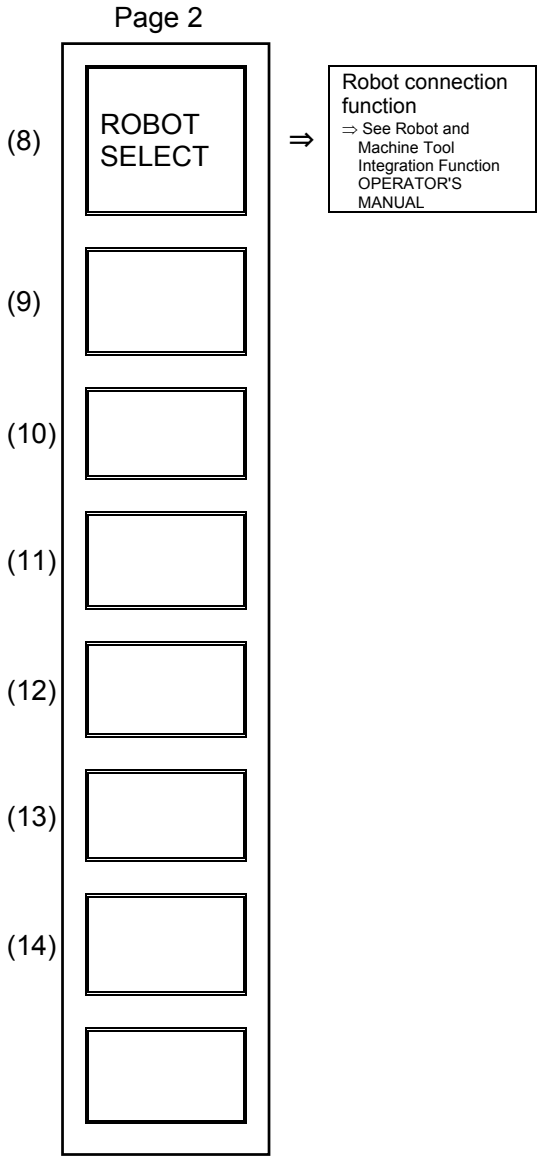
Screen displayed when the function key  is pressed (for 15/19-inch display unit)

Screen displayed when the function key PROG is pressed (for 8.4/10.4-inch display unit)

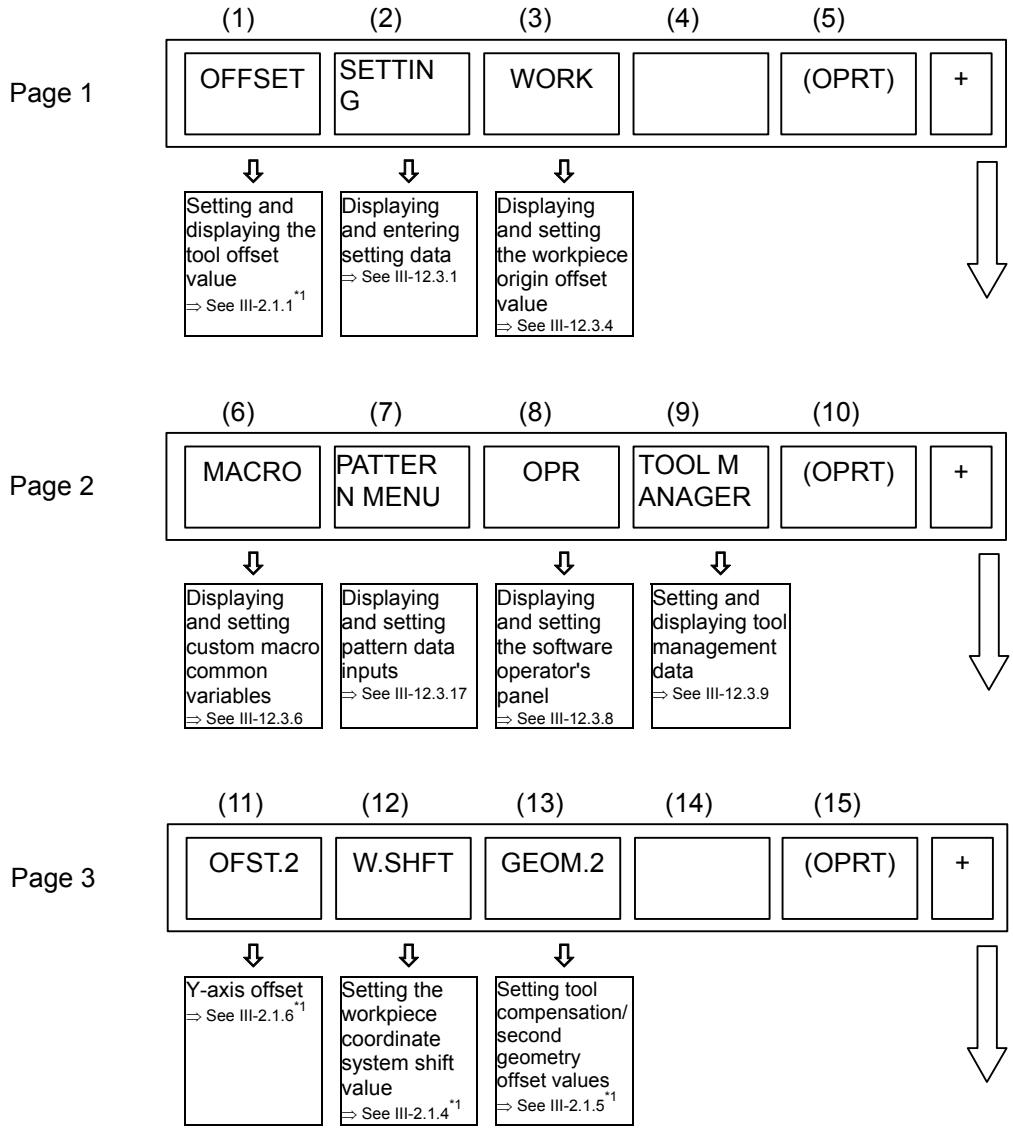


Screen displayed when the function key PROG is pressed (for 15/19-inch display unit)

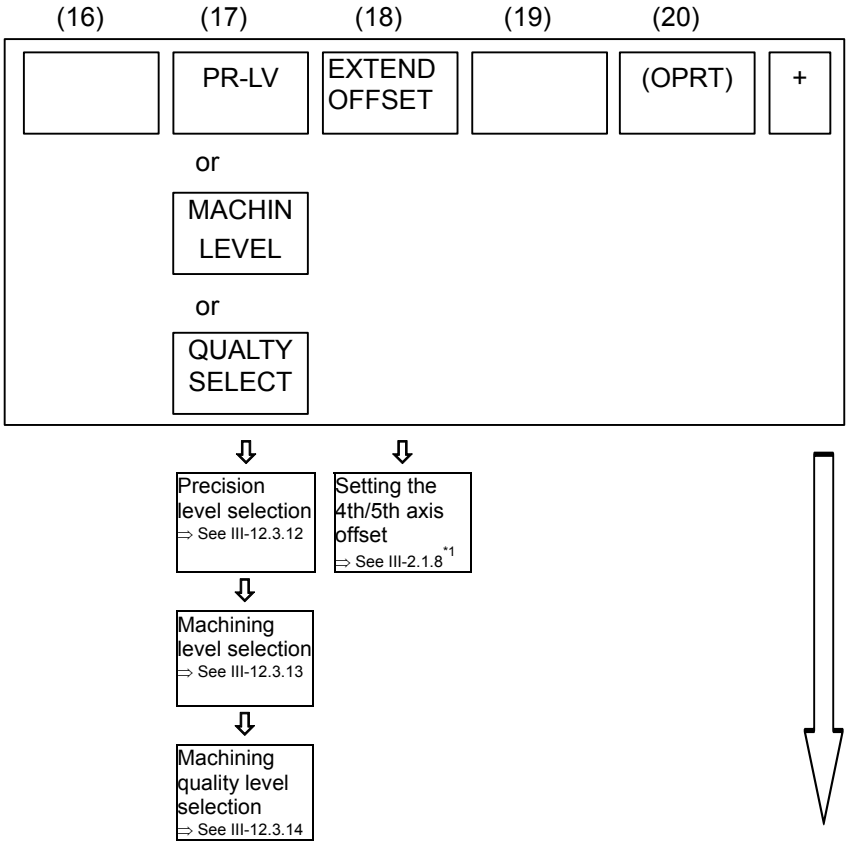



Page 2
(8)
ROBOT SELECT
⇒
Robot connection function
⇒ See Robot and Machine Tool Integration Function OPERATOR'S MANUAL
(9)
(10)
(11)
(12)
(13)
(14)

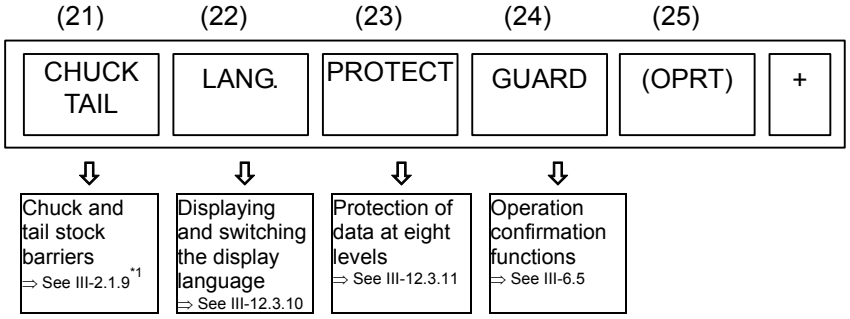
Screen displayed when the function key is pressed (for 8.4/10.4-inch display unit)



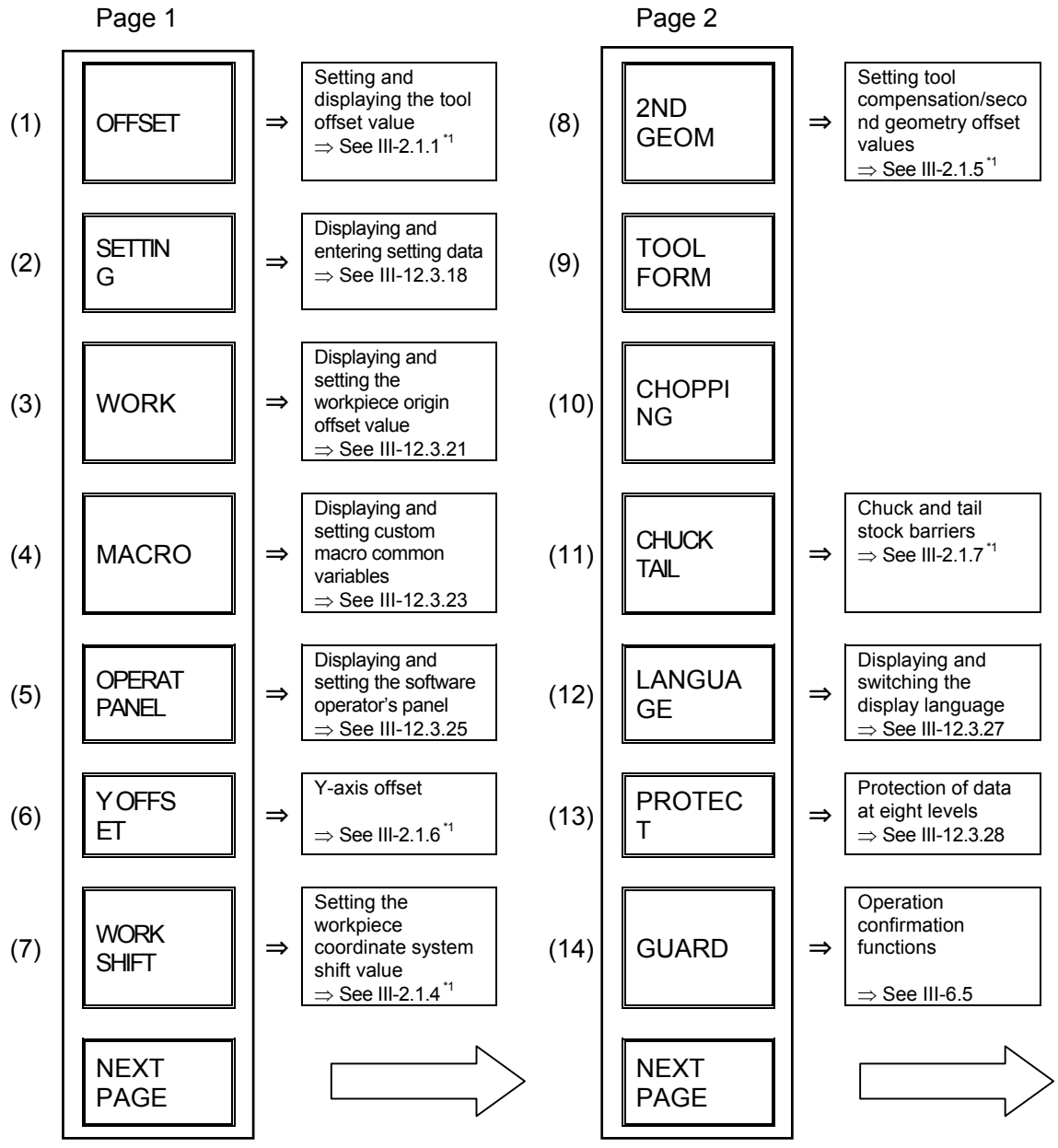
Page 4


(16)
(17)
(18)
(19)
(20)
PR-LV
EXTEND OFFSET
(OPRT)
+
or
MACHIN LEVEL
or
QUALTY SELECT
Precision level selection
⇨ See III-12.3.12
Setting the 4th/5th axis offset
⇨ See III-2.1.8 *1
Machining level selection
⇨ See III-12.3.13
Machining quality level selection
⇨ See III-12.3.14

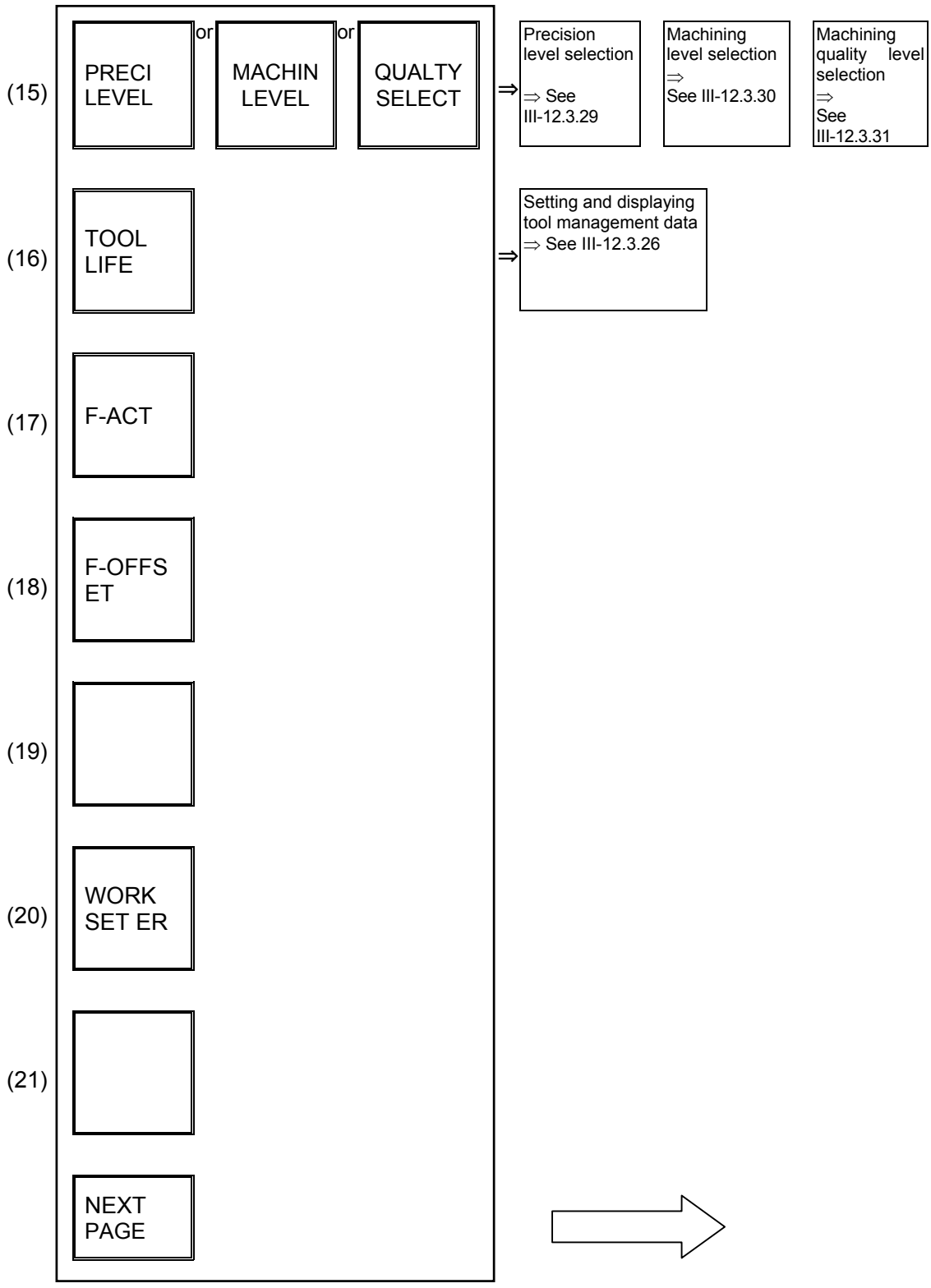
Page 5


(21)
(22)
(23)
(24)
(25)
CHUCK TAIL
LANG.
PROTECT
GUARD
(OPRT)
+
Chuck and tail stock barriers
⇨ See III-2.1.9 *1
Displaying and switching the display language
⇨ See III-12.3.10
Protection of data at eight levels
⇨ See III-12.3.11
Operation confirmation functions
⇨ See III-6.5

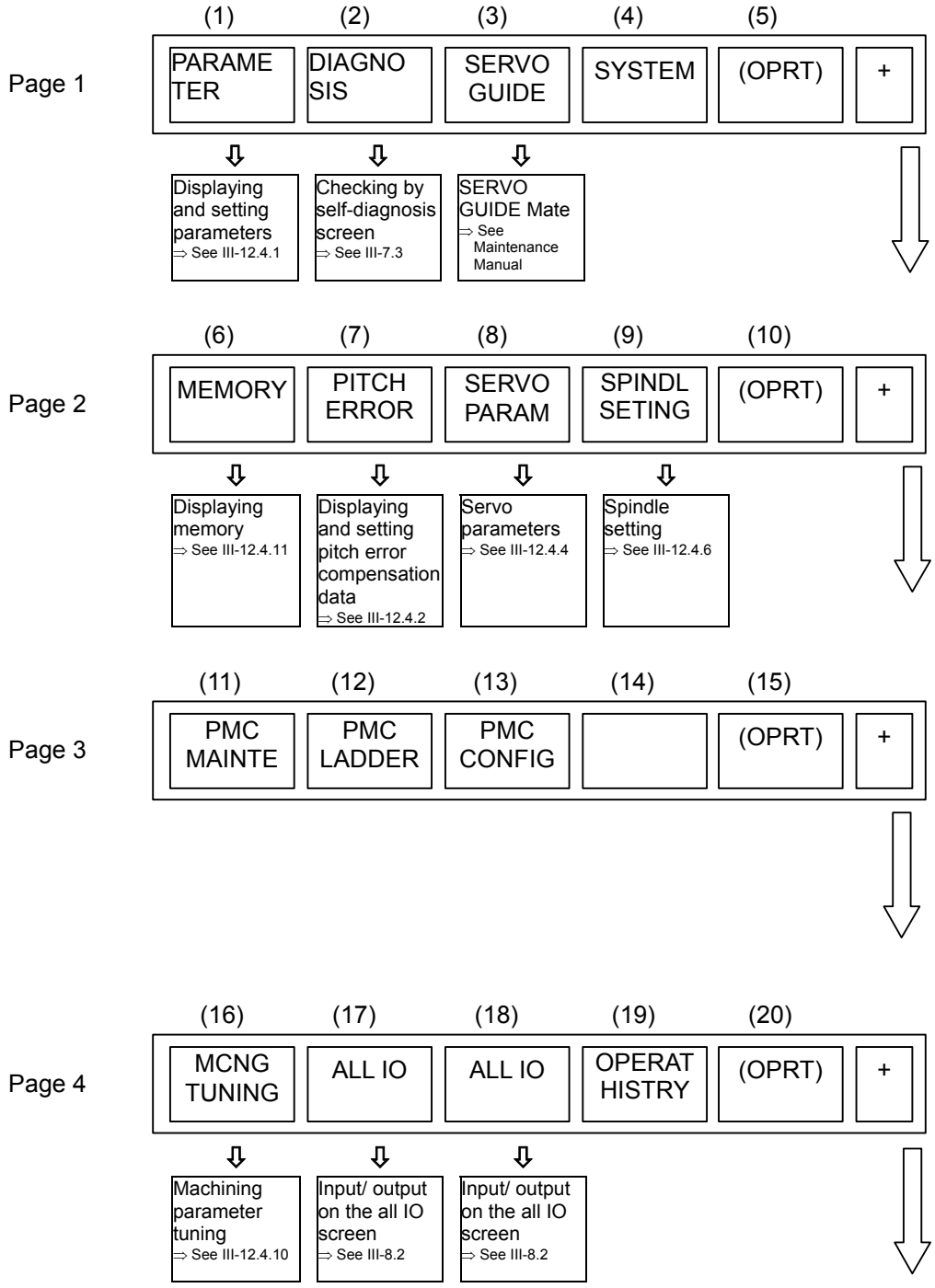
Screen displayed when the function key is pressed (for 15/19-inch display unit)



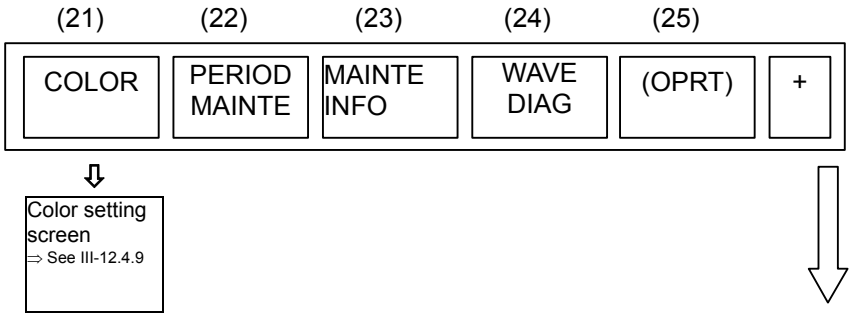
Page 3


(15)
PRECI LEVEL
or
MACHIN LEVEL
or
QUALTY SELECT
Precision level selection
⇒ See III-12.3.29
Machining level selection
⇒ See III-12.3.30
Machining quality level selection
⇒ See III-12.3.31
(16)
TOOL LIFE
Setting and displaying tool management data
⇒ See III-12.3.26
(17)
F-ACT
(18)
F-OFFSET
(19)
(20)
WORK SET ER
(21)
NEXT PAGE

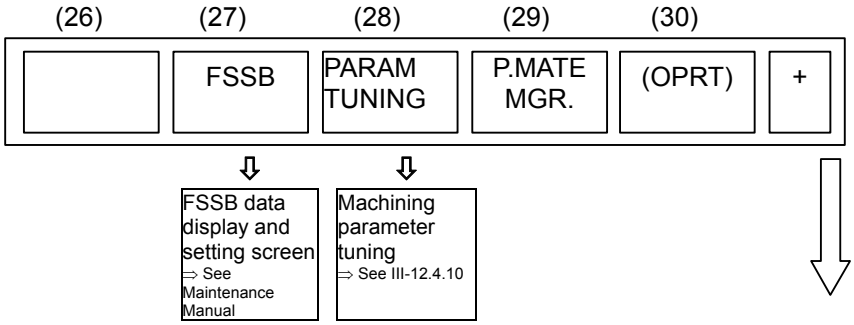
Screen displayed when the function key SYSTEM is pressed (for 8.4/10.4-inch display unit)



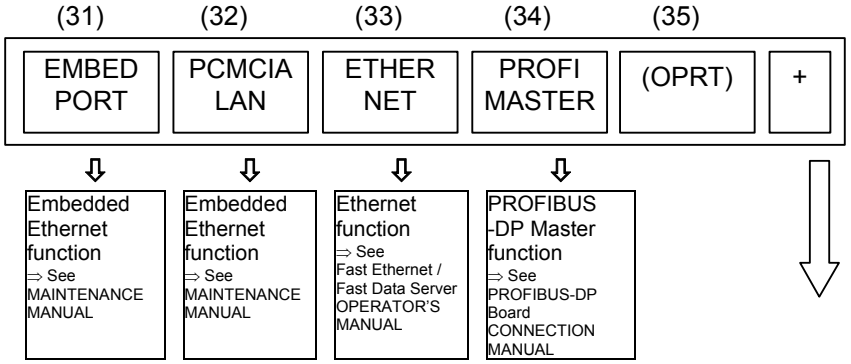
Page 5



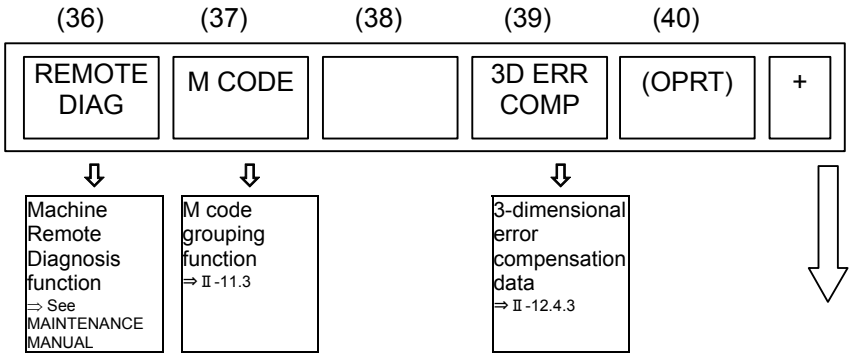
Page 6

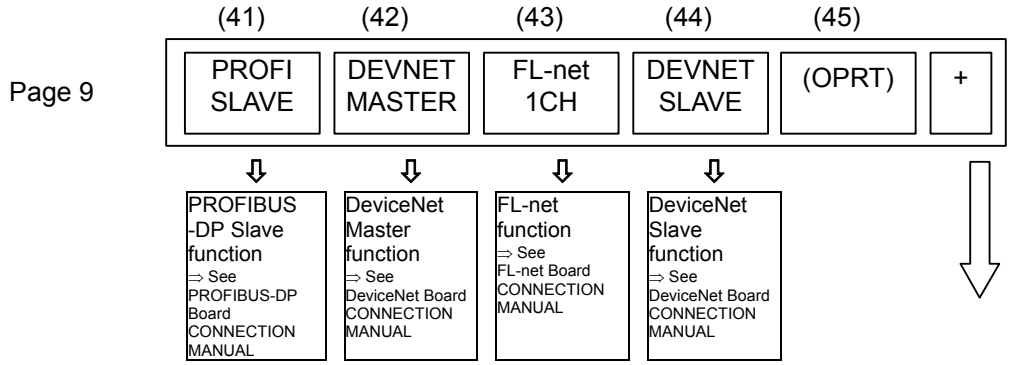


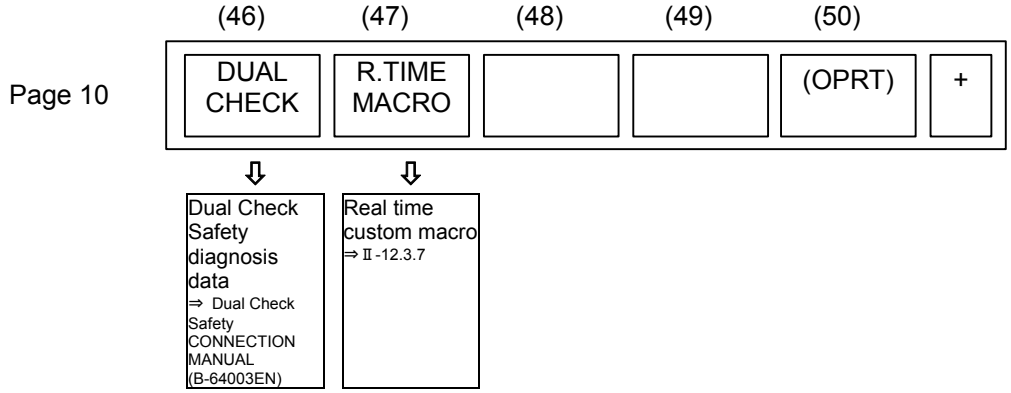
Page 7

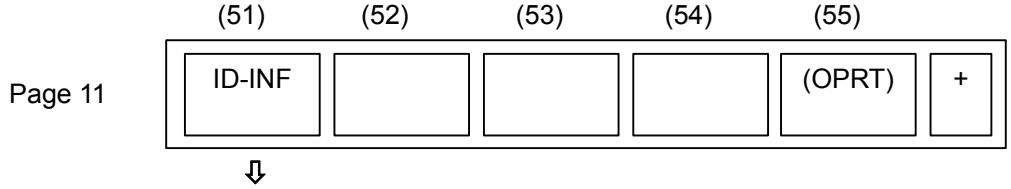


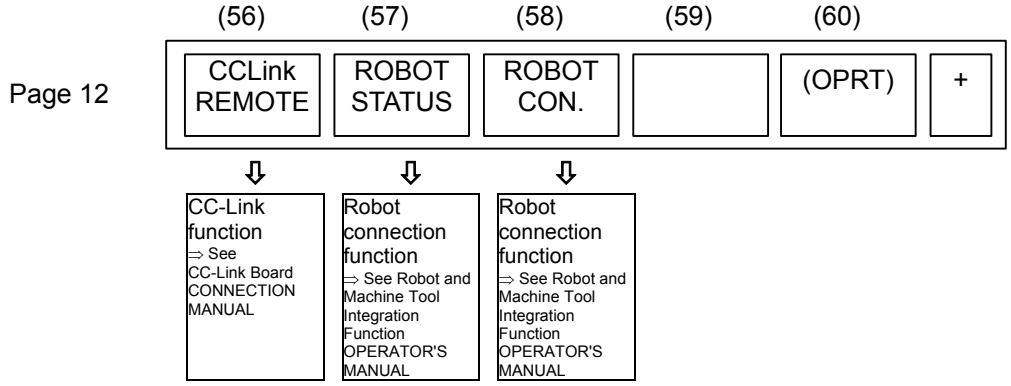
Page 8



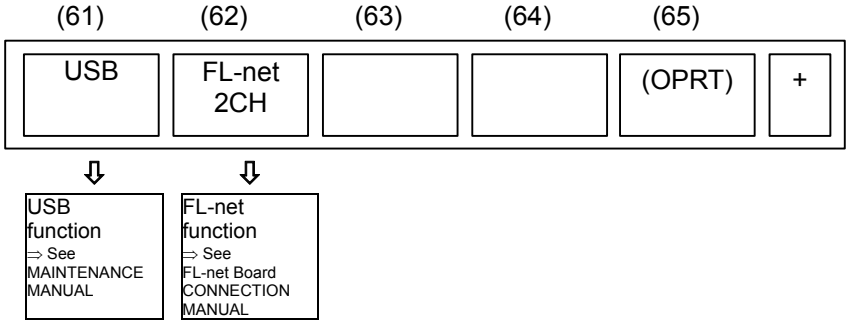

(41)
(42)
(43)
(44)
(45)
Page 9
PROFI SLAVE
DEVNET MASTER
FL-net 1CH
DEVNET SLAVE
(OPRT)
+
PROFIBUS -DP Slave function
⇒ See PROFIBUS-DP Board CONNECTION MANUAL
DeviceNet Master function
⇒ See DeviceNet Board CONNECTION MANUAL
FL-net function
⇒ See FL-net Board CONNECTION MANUAL
DeviceNet Slave function
⇒ See DeviceNet Board CONNECTION MANUAL


(46)
(47)
(48)
(49)
(50)
Page 10
DUAL CHECK
R.TIME MACRO
(OPRT)
+
Dual Check Safety diagnosis data
⇒ Dual Check Safety CONNECTION MANUAL (B-64003EN)
Real time custom macro
⇒ Ⅱ-12.3.7


(51)
(52)
(53)
(54)
(55)
Page 11
ID-INF
(OPRT)
+


(56)
(57)
(58)
(59)
(60)
Page 12
CCLink REMOTE
ROBOT STATUS
ROBOT CON.
(OPRT)
+
CC-Link function
⇒ See CC-Link Board CONNECTION MANUAL
Robot connection function
⇒ See Robot and Machine Tool Integration Function OPERATOR'S MANUAL
Robot connection function
⇒ See Robot and Machine Tool Integration Function OPERATOR'S MANUAL

Page 13



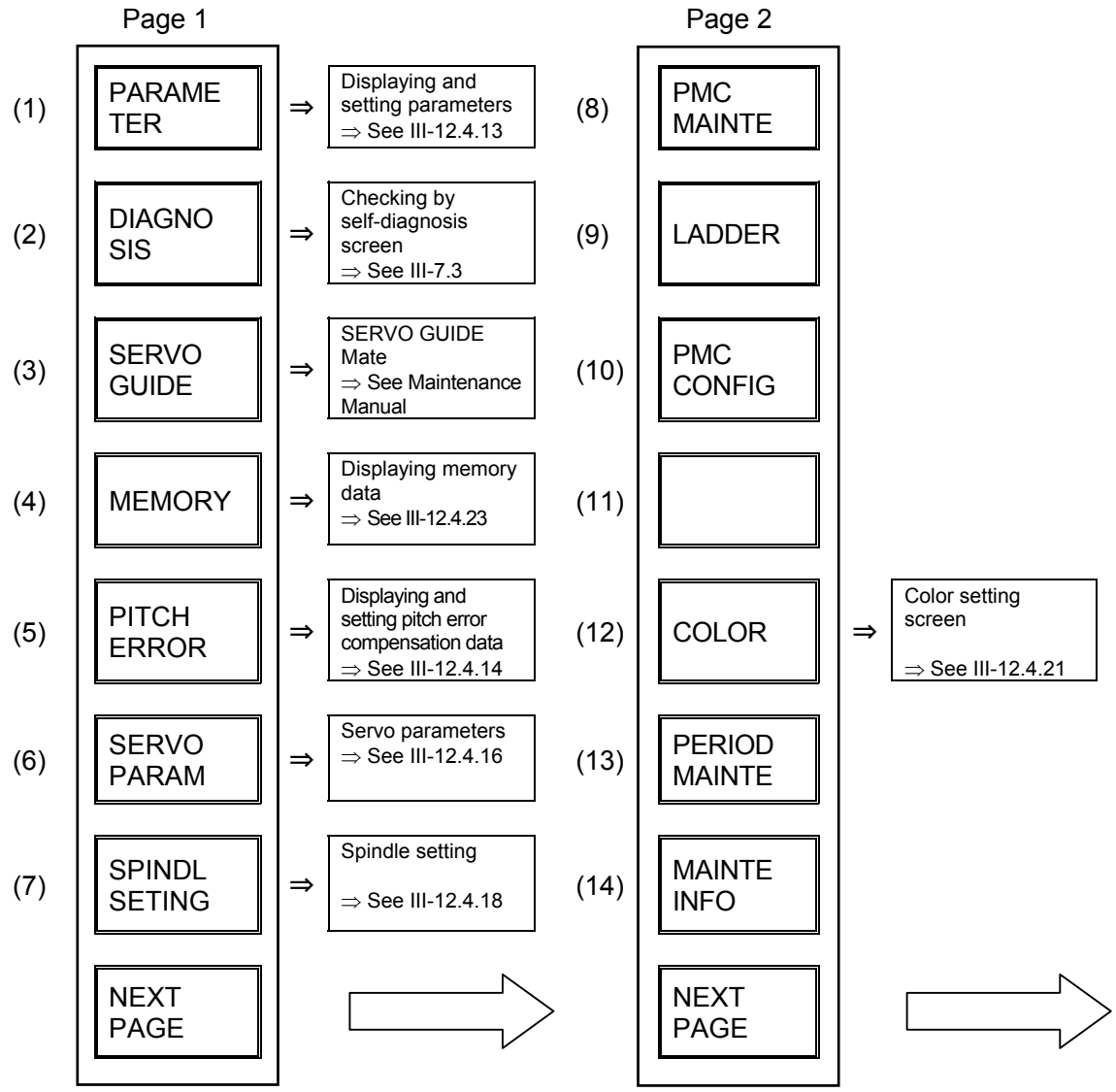
NOTE

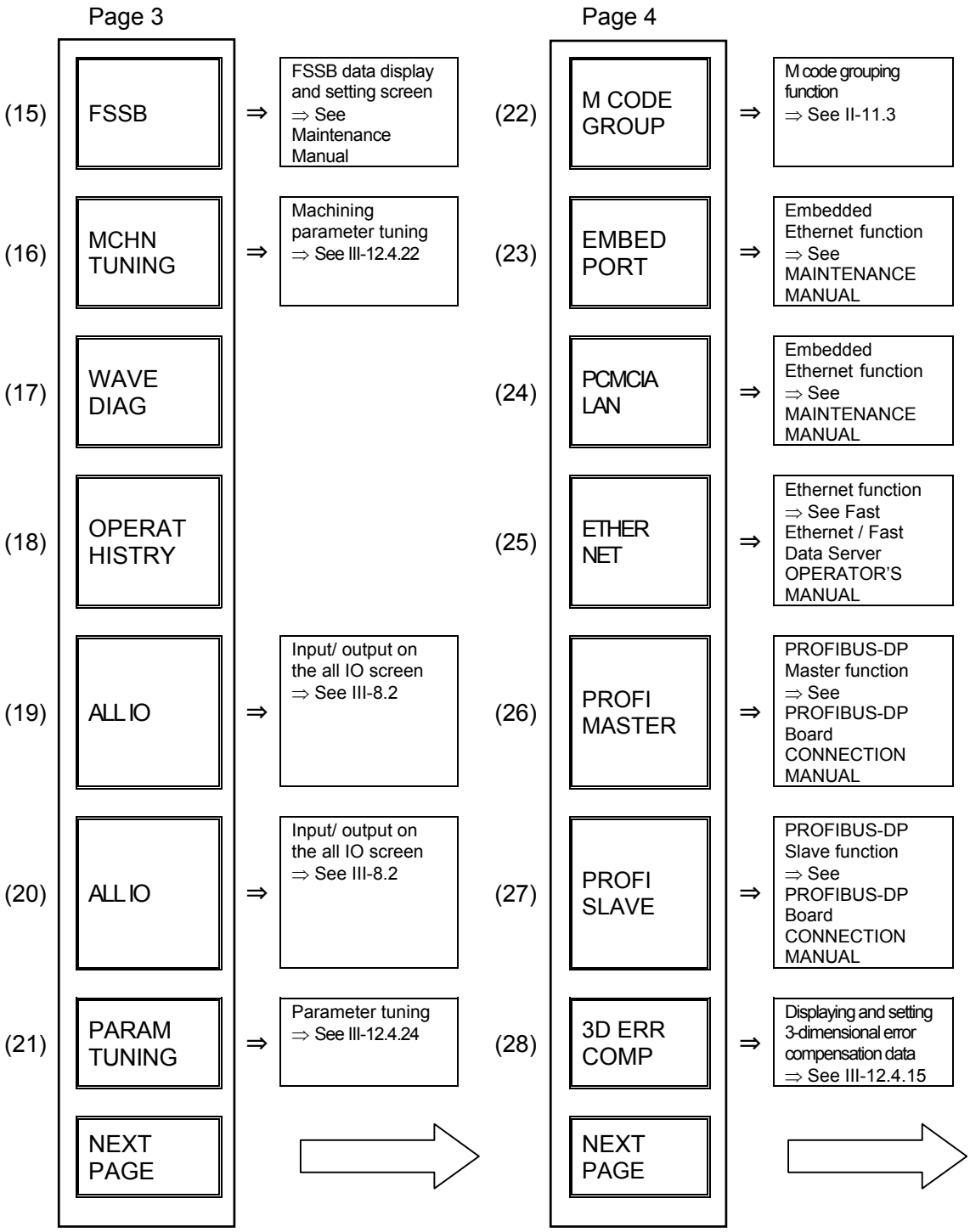
For information about a dedicated screen for each path control type in the lathe system/machining center system, refer to the manuals:

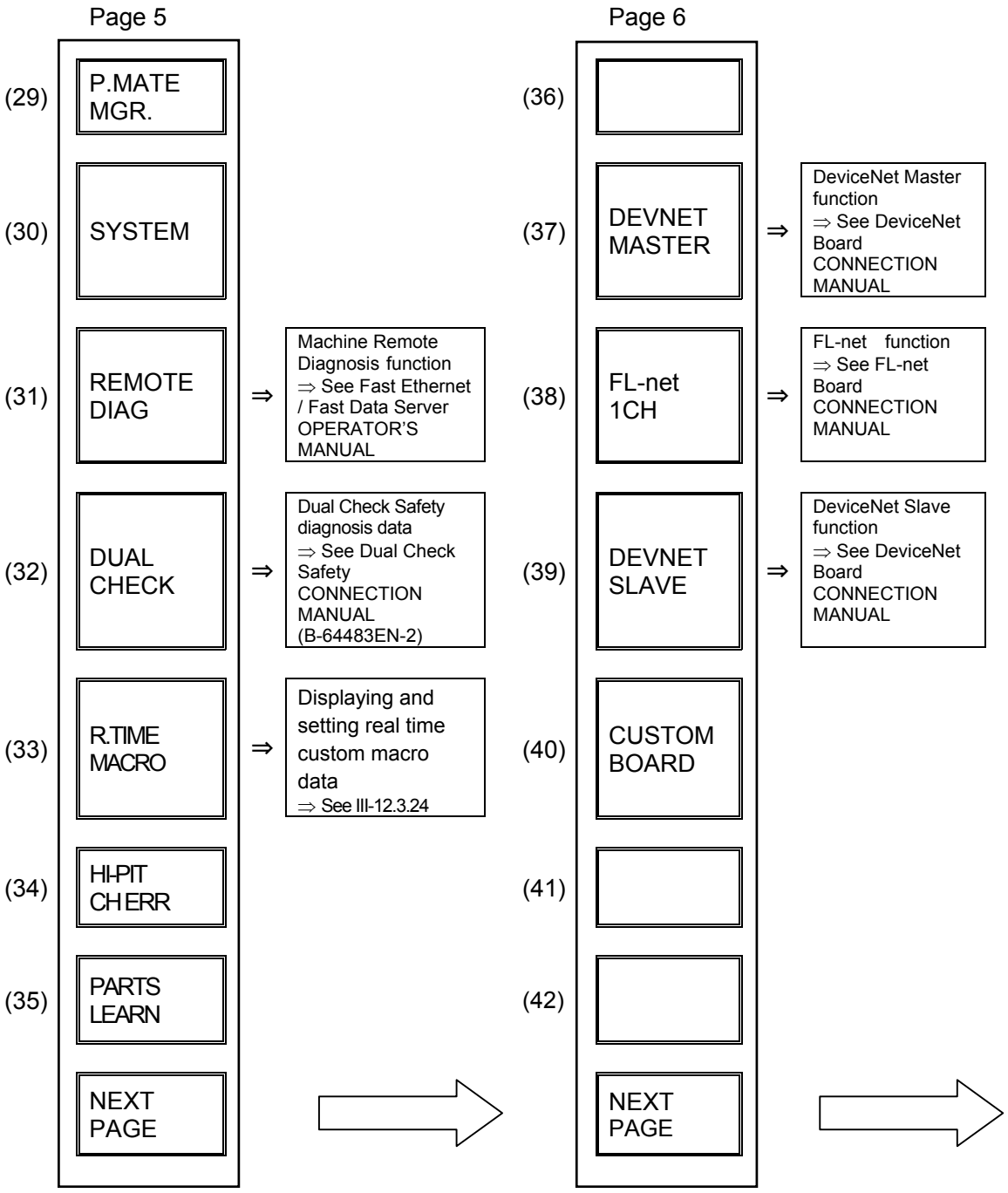
*1: OPERATOR'S MANUAL (T series) (B-64484EN-1)

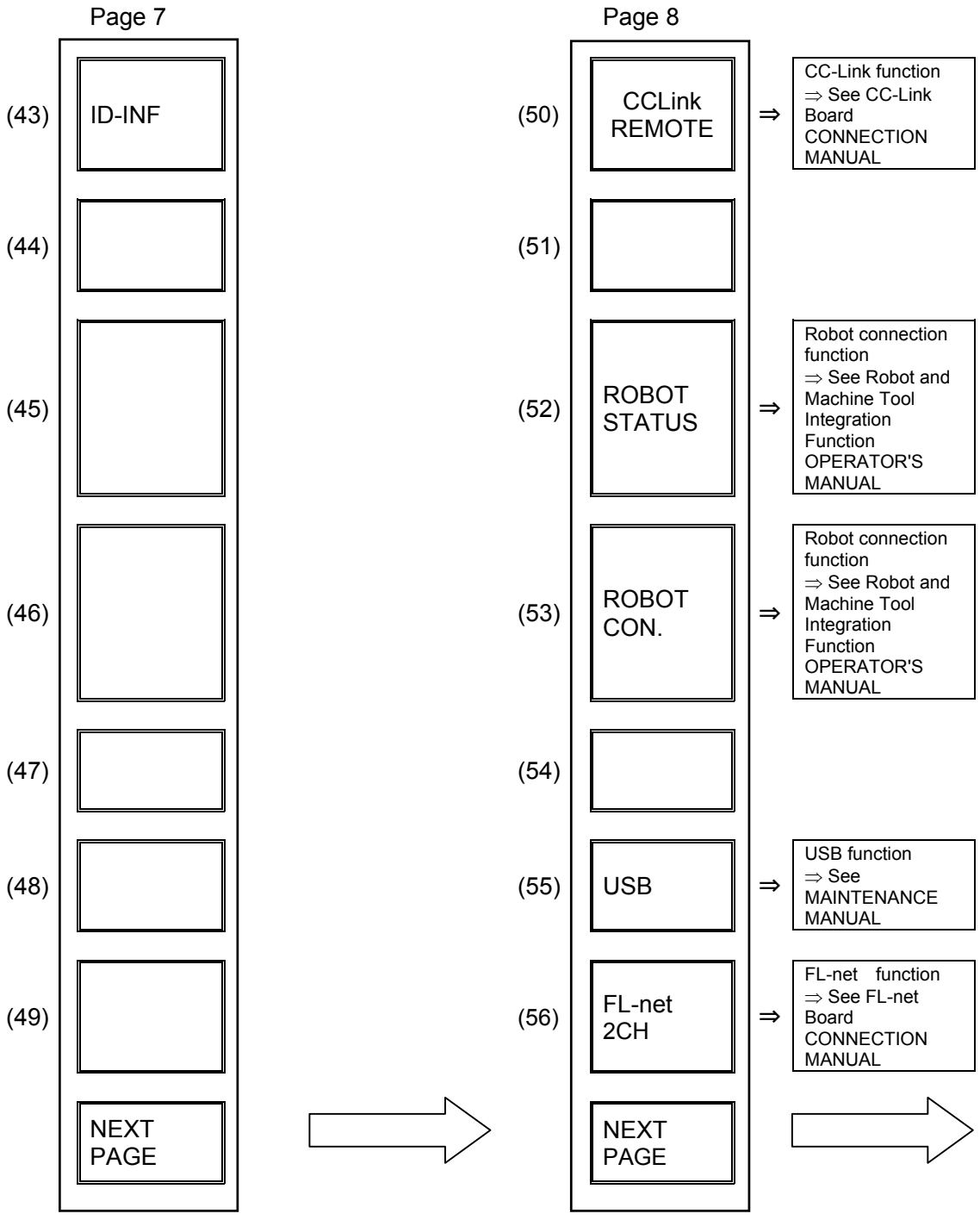
*2: OPERATOR'S MANUAL (M series) (B-64484EN-2)

Screen displayed when the function key SYSTEM is pressed (for 15/19-inch display unit)




Page 3
(15)
FSSB
⇒
FSSB data display and setting screen ⇒ See Maintenance Manual
(16)
MCHN TUNING
⇒
Machining parameter tuning ⇒ See III-12.4.22
(17)
WAVE DIAG
(18)
OPERAT HISTRY
(19)
ALL IO
⇒
Input/ output on the all IO screen ⇒ See III-8.2
(20)
ALL IO
⇒
Input/ output on the all IO screen ⇒ See III-8.2
(21)
PARAM TUNING
⇒
Parameter tuning ⇒ See III-12.4.24
NEXT PAGE
Page 4
(22)
M CODE GROUP
⇒
M code grouping function ⇒ See II-11.3
(23)
EMBED PORT
⇒
Embedded Ethernet function ⇒ See MAINTENANCE MANUAL
(24)
PCMCIA LAN
⇒
Embedded Ethernet function ⇒ See MAINTENANCE MANUAL
(25)
ETHER NET
⇒
Ethernet function ⇒ See Fast Ethernet / Fast Data Server OPERATOR'S MANUAL
(26)
PROFI MASTER
⇒
PROFIBUS-DP Master function ⇒ See PROFIBUS-DP Board CONNECTION MANUAL
(27)
PROFI SLAVE
⇒
PROFIBUS-DP Slave function ⇒ See PROFIBUS-DP Board CONNECTION MANUAL
(28)
3D ERR COMP
⇒
Displaying and setting 3-dimensional error compensation data ⇒ See III-12.4.15
NEXT PAGE


Page 5
(29)
P.MATE MGR.
(30)
SYSTEM
(31)
REMOTE DIAG
⇒
Machine Remote Diagnosis function
⇒ See Fast Ethernet / Fast Data Server OPERATOR'S MANUAL
(32)
DUAL CHECK
⇒
Dual Check Safety diagnosis data
⇒ See Dual Check Safety CONNECTION MANUAL (B-64483EN-2)
(33)
R.TIME MACRO
⇒
Displaying and setting real time custom macro data
⇒ See III-12.3.24
(34)
HI-PIT CH ERR
(35)
PARTS LEARN
NEXT PAGE
Page 6
(36)
(37)
DEVNET MASTER
⇒
DeviceNet Master function
⇒ See DeviceNet Board CONNECTION MANUAL
(38)
FL-net 1CH
⇒
FL-net function
⇒ See FL-net Board CONNECTION MANUAL
(39)
DEVNET SLAVE
⇒
DeviceNet Slave function
⇒ See DeviceNet Board CONNECTION MANUAL
(40)
CUSTOM BOARD
(41)
(42)
NEXT PAGE



NOTE

For information about a dedicated screen for each path control type in the lathe system/machining center system, refer to the manuals:
*1: OPERATOR'S MANUAL (T series) (B-64484EN-1)
*2: OPERATOR'S MANUAL (M series) (B-64484EN-2)

12.1 SCREENS DISPLAYED BY FUNCTION KEY

Section 12.1, "SCREENS DISPLAYED BY FUNCTION KEY", consists of the following subsections:

------ Screens of a 8.4/10.4-inch display unit ------
12.1.1 Position Display in the Workpiece Coordinate System....1344
12.1.2 Position Display in the Relative Coordinate System....1346
12.1.3 Overall Position Display....1349
12.1.4 Workpiece Coordinate System Preset....1350
12.1.5 Actual Feedrate Display....1351
12.1.6 Display of Run Time and Parts Count....1353
12.1.7 Setting the Floating Reference Position....1355
12.1.8 Operating Monitor Display....1355
12.1.9 Display of 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount)....1357
------ Screens of a 15/19-inch display unit ------
12.1.10 Overall Position Display (15/19-inch Display Unit)....1360
12.1.11 Workpiece Coordinate System Preset (15/19-inch Display Unit)....1363
12.1.12 Actual Feedrate Display (15/19-inch Display Unit)....1363
12.1.13 Display of Run Time and Parts Count (15/19-inch Display Unit)....1365
12.1.14 Setting the Floating Reference Position (15/19-inch Display Unit)....1367
12.1.15 Operating Monitor Display (15/19-inch Display Unit)....1367
12.1.16 Display of 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount) (15/19-inch Display Unit)....1369

Screens of a 8.4/10.4-inch display unit

Press function key POS to display the current position of the tool.

The following three screens are used to display the current position of the tool:

- Current position display screen for the workpiece coordinate system.
- Current position display screen for the relative coordinate system.
- Current overall position display screen.

The above screens can also display the feedrate, run time, and the number of parts. In addition, a floating reference position can be set on these screens.

Function key POS can also be used to display the load on the servo motor and spindle motor and the rotation speed of the spindle motor (operating monitor display).

Function key POS can also be used to display the screen for displaying the distance moved by handle interruption. See Section, “MANUAL HANDLE INTERRUPTION” for details on this screen.

On any of the position display screens, the status (such as D, I, L, S, *, or M) of an axis is indicated on the left of the axis name to prevent wrong operations. See "Axis status display" in III-6.5, "WRONG OPERATION PREVENTION FUNCTIONS" for details.

12.1.1 Position Display in the Workpiece Coordinate System

Displays the current position of the tool in the workpiece coordinate system. The current position changes as the tool moves. The least input increment is used as the unit for numeric values. The title at the top of the screen indicates that absolute coordinates are used.

Display procedure for the current position screen in the workpiece coordinate

Procedure

1 Press function key .

2 Press soft key [ABSOLUTE].

3 Press soft key [ABSOLUTE] again. The sixth and subsequent axes are displayed.

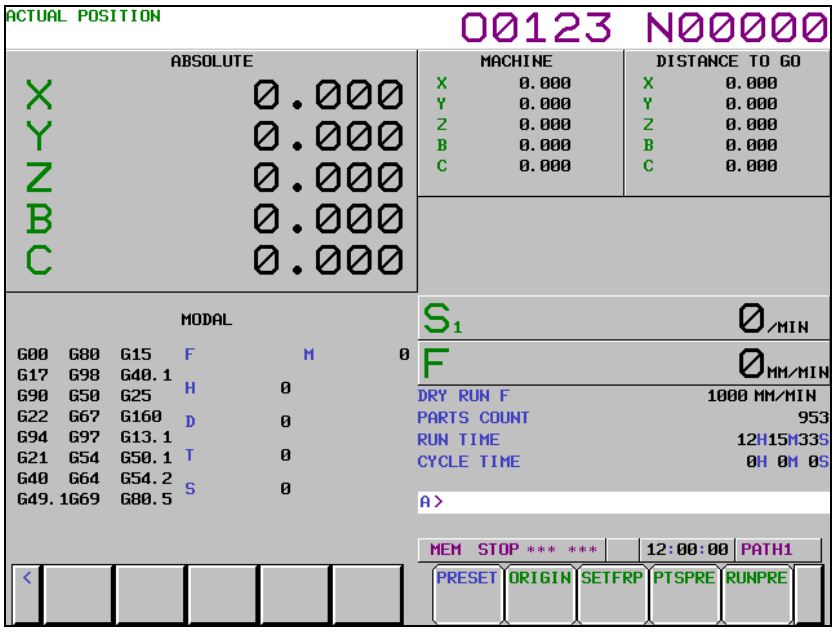


Fig. 12.1.1 (a) Current position (absolute) screen (M series)(10.4-inch display unit)

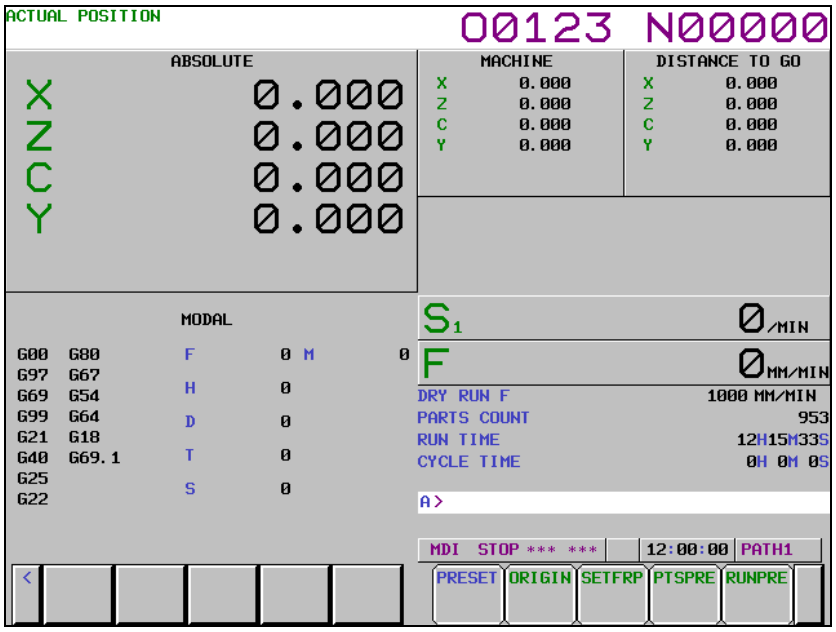


Fig. 12.1.1 (b) Current position (absolute) screen (T series)(10.4-inch display unit)

Explanation

- Display including compensation values

M

Bits 6 (DAL) and 7 (DAC) of parameter No. 3104 can be used to select whether the displayed values include tool length compensation and cutter compensation.

T

Bit 1 (DAP) parameter No. 3129 and bit 7 (DAC) of parameter No. 3104 can be used to select whether the displayed values include tool offset and tool nose radius compensation.

12.1.2 Position Display in the Relative Coordinate System

Displays the current position of the tool in a relative coordinate system based on the coordinates (see Explanation) set by the operator. The current position changes as the tool moves. The increment system is used as the unit for numeric values.
The title at the top of the screen indicates that relative coordinates are used.

Display procedure for the current position screen with the relative coordinate system

Procedure

1 Press function key 

.

2 Press soft key [RELATIVE].

3 Press soft key [RELATIVE] again. The sixth and subsequent axes are displayed.

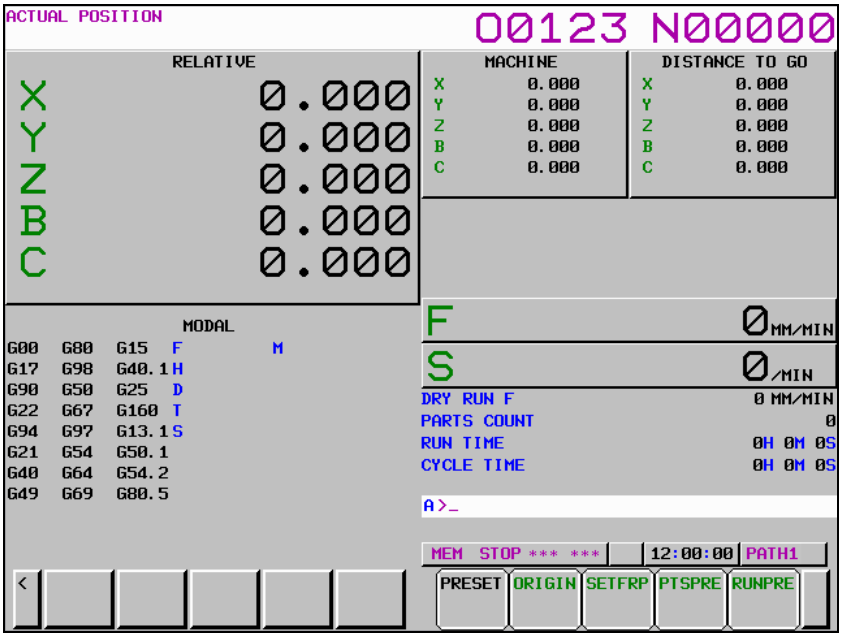


Fig. 12.1.2 (a) Current position (relative) screen (M series)(10.4-inch display unit)

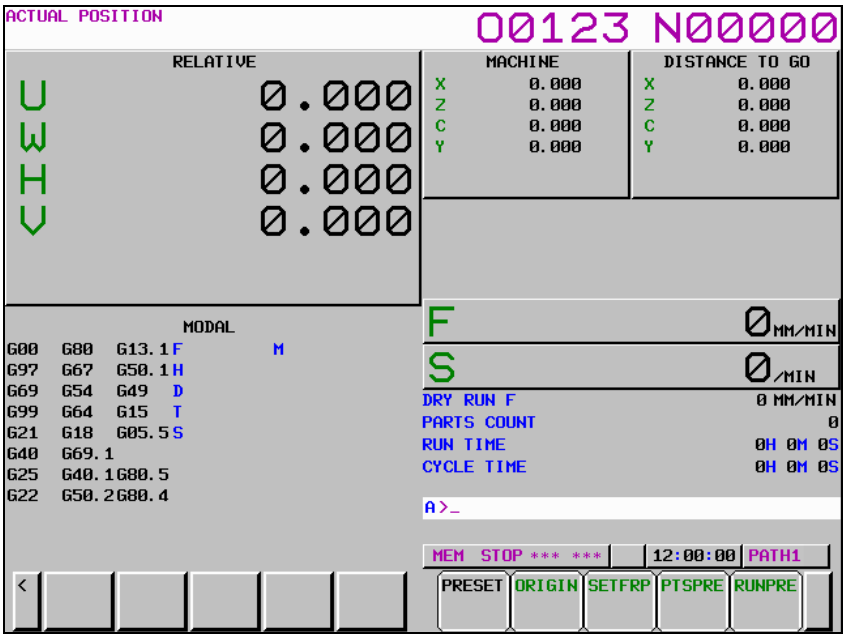


Fig. 12.1.2 (b) Current position (relative) screen (T series) (10.4-inch display unit)

See Explanation for the procedure for setting the coordinates.

Explanation

- Setting the relative coordinates

The current position of the tool in the relative coordinate system can be reset to 0 or preset to a specified value as follows.

Normal decimal point input and Pocket calculator type decimal point input can be selected by bit 0 (DPI) of parameter.No.3401.

Resetting relative coordinates to 0

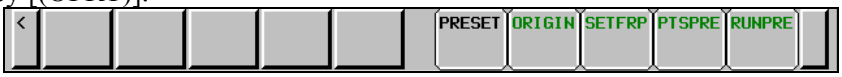
When all axes are reset to 0

1 Press function key 

.

2 Press chapter selection key [RELATIVE] to display the relative coordinate screen.

3 Press soft key [(OPRT)].



4 Press soft key [ORIGIN].



5 Press soft key [ALL AXIS].
The relative coordinates of all axes are reset to 0.

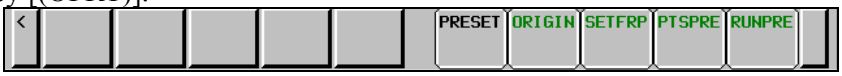
When a specified axis is reset to 0

1 Press function key 

.

2 Press chapter selection key [RELATIVE] to display the relative coordinate screen.

3 Press soft key [(OPRT)].



4 Press soft key [ORIGIN].



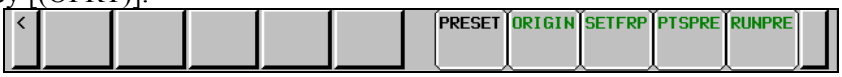
5 Input the axis name of the resetting axis.
The axis name blinks.
6 Press soft key [EXEC].
The relative coordinate is reset to 0.

Presetting relative coordinates

Procedure

1 Press function key 

.
2 Press chapter selection key [RELATIVE] to display the relative coordinate screen.
3 Press soft key [(OPRT)].



4 Input the axis name of the presetting axis.
The axis name blinks.
5 Input the coordinates value, and press soft key [PRESET].
The relative coordinate is preset by input coordinate value.

NOTE
1 When the preset of the specified axis is done in the following states, warning is displayed.
- When only the numerical value is input or the character that is not the axis name is input
  Waning massage: “THERE IS NO SPECIFIED NAME OF AXIS”
- When input form is except for “axis name + numerical value” is done
  Waning massage: ”FORMAT ERROR”

2 When axis name display is changed by parameter No.3132, specify the changed axis name.

- Display including compensation values

M

Bits 4 (DRL) and 5 (DRC) of parameter No. 3104 can be used to select whether the displayed values include tool length compensation and cutter compensation.

T

Bit 0 (DRP) parameter No. 3129 and bit 5 (DRC) of parameter No. 3104 can be used to select whether the displayed values include tool offset and tool nose radius compensation.

- Presetting by setting a coordinate system

M

Bit 3 (PPD) of parameter No. 3104 can be used to specify whether the position indication values in the absolute coordinate system are preset as those in the relative coordinate system during coordinate system setting or manual reference position return by the G92 command.

T

Bit 3 (PPD) of parameter No. 3104 can be used to specify whether the position indication values in the absolute coordinate system are preset as those in the relative coordinate system during coordinate system setting or manual reference position return by the G50 command (for G code system A) or G92 command (for G code system B or C).

If parameter bit PPD is set to 0, by setting bit 5 (PWR) of parameter No. 11277 to 1, it is possible to preset the relative coordinates at power-on in the machine coordinate system.

12.1.3 Overall Position Display

Displays the following positions on a screen : Current positions of the tool in the workpiece coordinate system, relative coordinate system, and machine coordinate system, and the remaining distance.
The relative coordinates can also be set on this screen. See Subsection, “Position Display in the Relative Coordinate System” for the procedure.

Procedure for displaying overall position display screen

Procedure

1 Press function key 

.

2 Press soft key [ALL].

3 Press soft key [ALL] again. The sixth and subsequent axes are displayed.

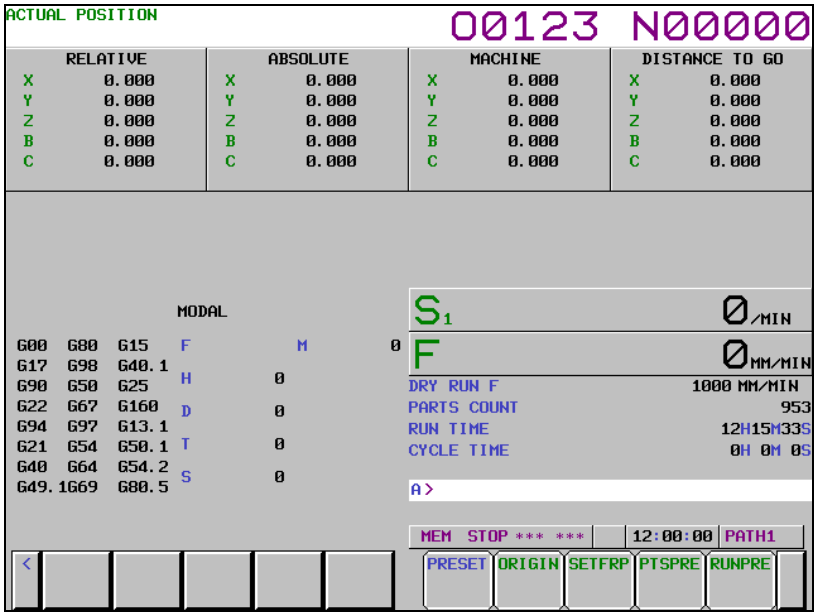


Fig. 12.1.3 (a) Current position (overall) screen (M series) (10.4-inch display unit)

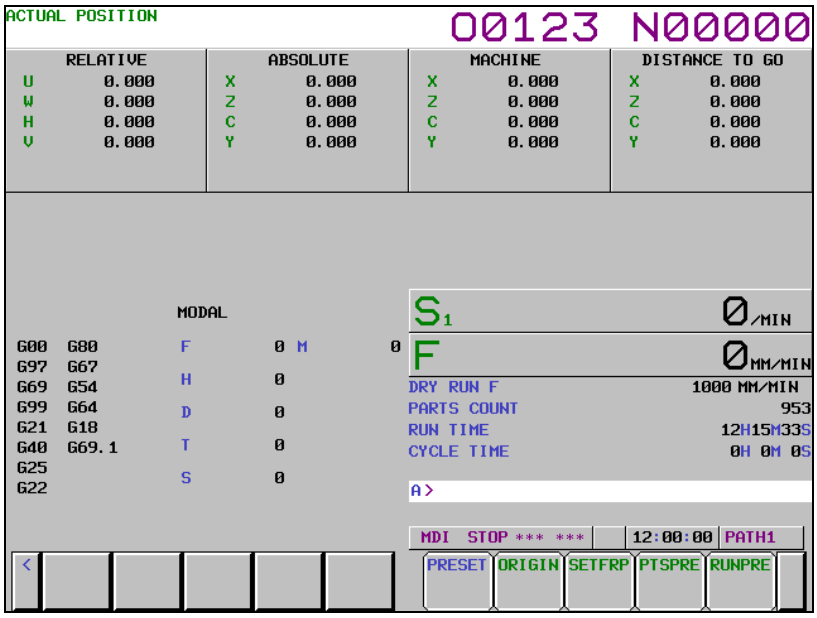


Fig. 12.1.3 (b) Current position (overall) screen (T series) (10.4-inch display unit)

Explanation

- Coordinate display

The current positions of the tool in the following coordinate systems are displayed at the same time:

- Current position in the relative coordinate system (relative coordinate)
- Current position in the workpiece coordinate system (absolute coordinate)
- Current position in the machine coordinate system (machine coordinate)
- Distance to go (distance to go)

- Distance to go

The distance remaining is displayed in the MEM or MDI mode.
The distance the tool is yet to be moved in the current block is displayed.

- Machine coordinate system

The least command increment is used as the unit for values displayed in the machine coordinate system. However, the least input increment can be used by setting bit 0 (MCN) of parameter No. 3104.

- Resetting the relative coordinates

The total position display screen also supports the resetting of the relative coordinates to 0 or presetting of them to specified values. See the procedure for resetting the relative coordinates described in See Subsection, “Position Display in the Relative Coordinate System”.

12.1.4 Workpiece Coordinate System Preset

If a workpiece coordinate system has been shifted with manual intervention or any other operation, an MDI operation can be performed to preset the system to a workpiece coordinate system that is offset by a workpiece origin offset, from the machine zero point that has been set before the shifting.
A command (G92.1) can be programmed to preset a workpiece coordinate system. (See II-7.2.4.)

Procedure for the workpiece coordinate system preset

Procedure

1 Press function key .

2 Press soft key [(OPRT)].

3 Press soft key [ALL] or [RELATIVE].

4 Enter the axis name (X , & Y , ...) and zero (P 0). (Fig. 12.1.4 (a))

5 Press soft key [PRESET].

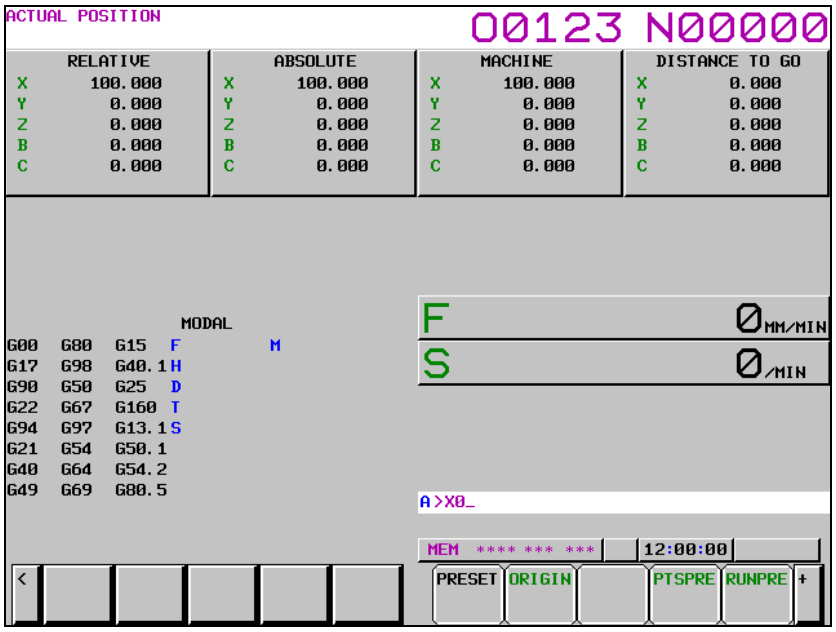


Fig. 12.1.4 (a) Overall screen

Explanation

- Operation mode

This function can be executed when the reset state or automatic operation stop state is entered, regardless of the operation mode.

- Presetting relative coordinates

As with absolute coordinates, bit 3 (PPD) of parameter No. 3104 is used to specify whether to preset relative coordinates (RELATIVE).

12.1.5 Actual Feedrate Display

The actual feedrate on the machine (per minute) can be displayed on a current position display screen or program check screen.
On the 12 soft keys display unit, the actual feedrate is always displayed.

Display procedure for the actual feedrate on the current position display screen

Procedure

1 Press the function key POS to display a current position display screen. At the location indicated by ⬚, an actual feedrate is displayed.

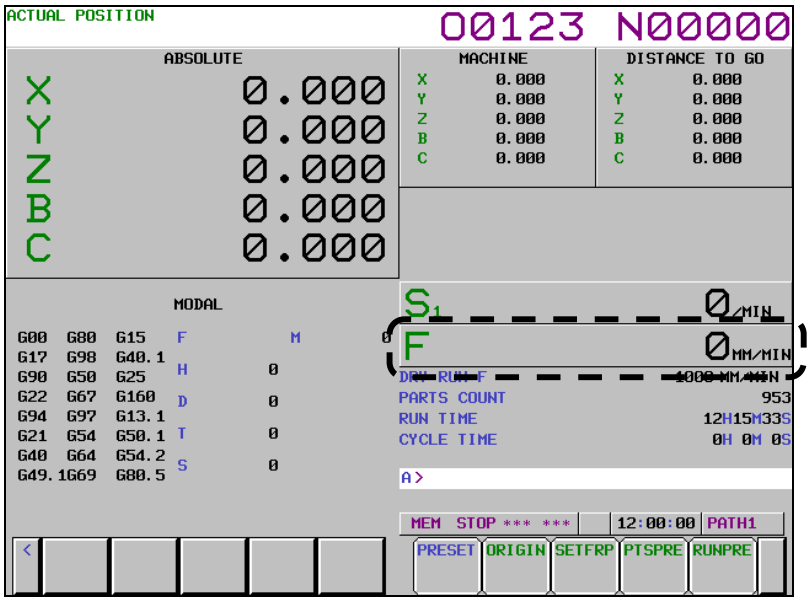


Fig. 12.1.5 (a) Current position (absolute) screen (M series) (10.4-inch display unit)

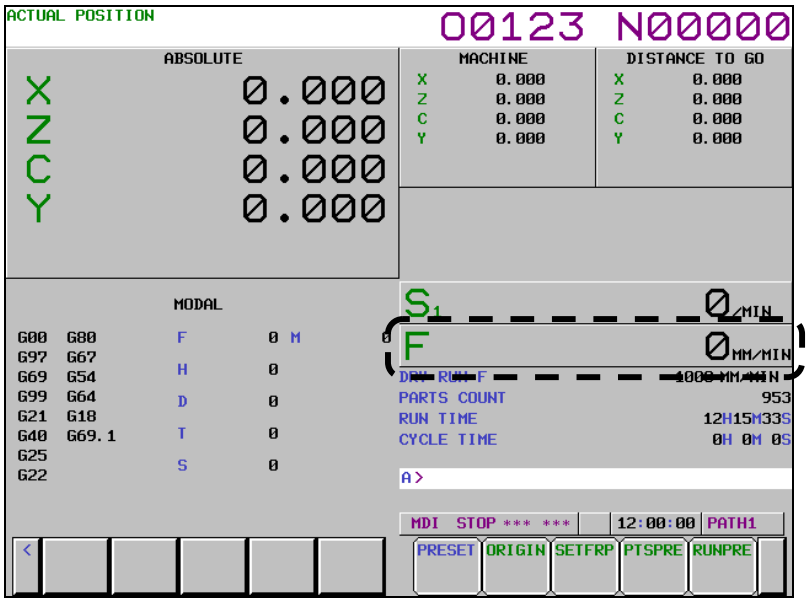


Fig. 12.1.5 (b) Current position (absolute) screen (T series) (10.4-inch display unit)

The actual feedrate is displayed in units of millimeter/min or inch/min (depending on the specified least input increment) under the display of the current position.

Explanation

- Actual feedrate value

The actual rate is calculated by the following expression:

$$Fact = \sqrt{\sum_{i=1}^{n} (fi)^2}$$

n : Number of axes
fi : Cutting feed rate in the tangential direction of each axis or rapid traverse rate
Fact : Actual feedrate displayed

The display unit:

mm/min (metric input).
inch/min (Inch input, two digits below the decimal point are displayed.)

- Actual feedrate display of feed per revolution

In the case of feed per revolution and thread cutting, the actual feedrate displayed is the feed per minute rather than feed per revolution.

- Actual feedrate display of rotary axis

In the case of movement of rotary axis, the speed is displayed in units of deg/min but is displayed on the screen in units of input system at that time. For example, when the rotary axis moves at 50 deg/min, the following is displayed: 50 mm/min (in metric input) or 0.50 inch/min (in inch input).

- Actual feedrate display on the other screen

The program check screen also displays the actual feedrate.

12.1.6 Display of Run Time and Parts Count

The run time, cycle time, and the number of machined parts are displayed on the current position display screens.

Procedure for displaying run time and parts count on the current position display screen

Procedure

1 Press the function key 

to display a current position display screen. At the location indicated by ⬚, a run time and parts count are displayed.

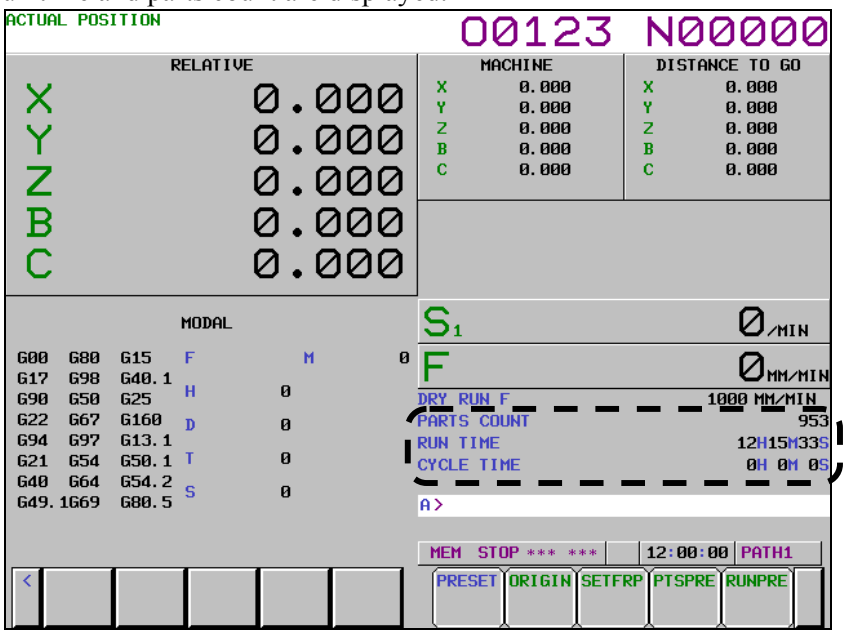


Fig. 12.1.6 (a) Current position (relative) screen (M series) (10.4-inch display unit)

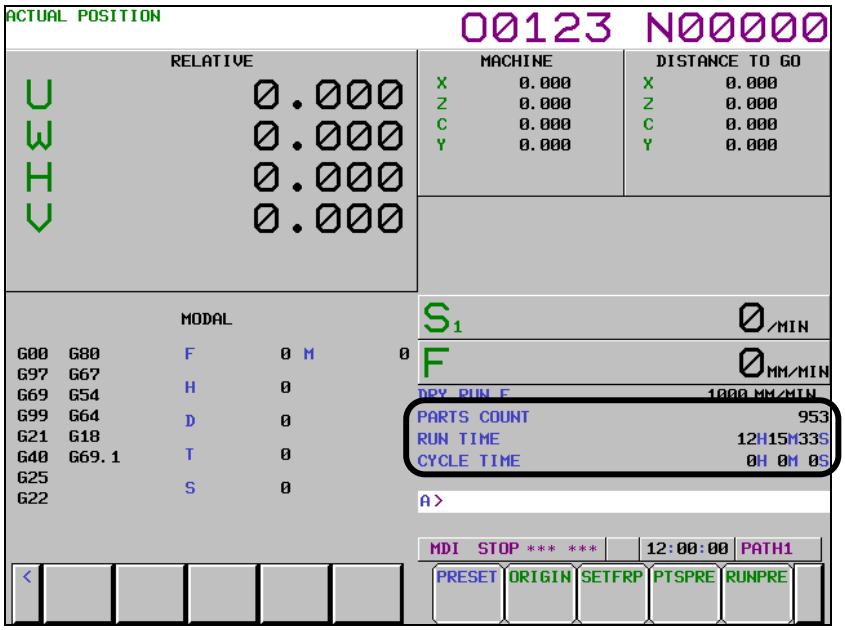


Fig. 12.1.6 (b) Current position (relative) screen (T series) (10.4-inch display unit)

The number of machined parts (PART COUNT), run time (RUN TIME), and cycle time (CYCLE TIME) are displayed under the current position.

Explanation

- PART COUNT

Indicates the number of machined parts. The number is incremented each time M02, M30, or an M code specified by parameter No. 6710 is executed.

- RUN TIME

Indicates the total run time during automatic operation, excluding the stop and feed hold time.

- CYCLE TIME

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- Display on the other screen

Details of the run time and the number of machined parts are displayed on the setting screen. See III-12.3.3.

- Parameter setting

The number of machined parts and run time cannot be set on current position display screens.
The number of machined parts and run time cannot be set on current position display screens. They can be set by parameters Nos. 6711, 6751, and 6752 or on the setting screen.

- Incrementing the number of machined parts

Bit 0 (PCM) of parameter No. 6700 is used to specify whether the number of machined parts is incremented each time M02, M30, or an M code specified by parameter No. 6710 is executed, or only each time an M code specified by parameter No. 6710 is executed.

12.1.7 Setting the Floating Reference Position

To perform floating reference position return with a G30.1 command, the floating reference position must be set beforehand.

Procedure for setting the floating reference position

Procedure

1 Press function key  to display a screen used for displaying the current position.
2 Move the tool to the floating reference position by jogging.
3 Press soft key [(OPRT)].
4 Press soft key [SET FRP].
5 To register the floating reference positions for all axes, press soft key [ALL AXIS].
To register the floating reference position of a specific axis, enter the name of the axis (, , etc.), then press soft key [AXIS NAME]. Two or more names can be entered consecutively (e.g.,)   [AXIS NAME]).
The above operation stores the floating reference position. It can be checked with parameter No. 1244.
6 In step 4, the floating reference position along a specified axis can also be stored by entering the axis name (such as  or ) and pressing soft key [SET FRP].

Explanation

- Presetting the relative coordinate system

When bit 3 (FPC) of parameter No. 1201 is set to 1, the floating reference position can be registered, while the relative position display can be preset to 0.

12.1.8 Operating Monitor Display

The load meter for a servo axis can be displayed. Also, the load meter and speed meter for a serial spindle can be displayed.
To enable this function, bit 5 (OPM) of parameter No. 3111 must be set to 1.

Procedure for displaying the operating monitor

Procedure

1 Press function key .
2 Press continuous menu key .
3 Press soft key [MONITOR].

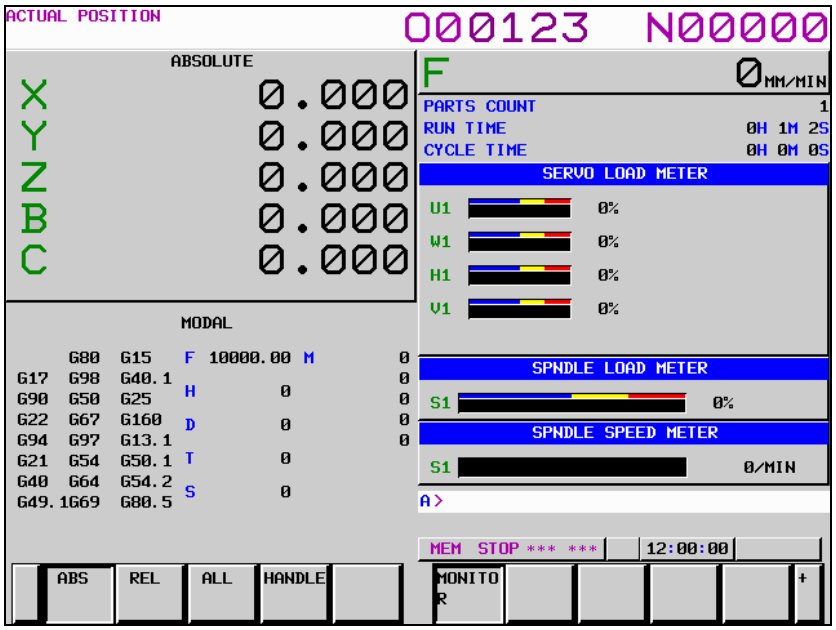


Fig. 12.1.8 (a) Operating monitor (M series) (10.4-inch display unit)

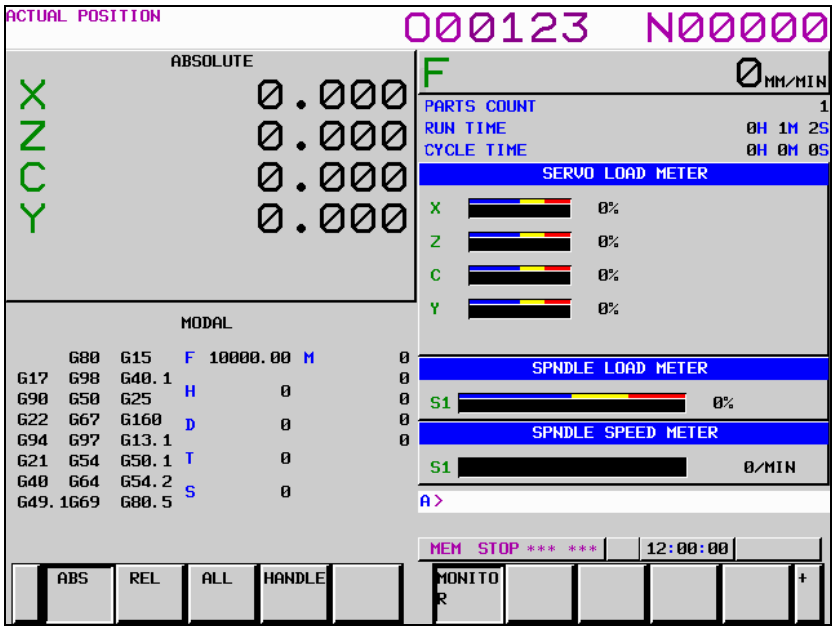


Fig. 12.1.8 (b) Operating monitor (T series) (10.4-inch display unit)

Explanation

- Display of the servo axes

Servo axis load meters as many as the maximum number of controlled axes of the path can be displayed. One screen displays load meters for up to five axes at a time. By pressing the soft key [MONITOR], load meters for the 6th axis and up are displayed.

- Display of the spindle axes

When serial spindles are used, the reading on the load meter and speedometer can be displayed only for the main serial spindle.

- Unit of graph

The bar graph for the load meter shows load up to 200% (only a value is displayed for load exceeding 200%).

The bar graph for the speedometer shows the ratio of the current spindle speed to the maximum spindle speed (100%).

- Load meter

The reading on the load meter depends on servo parameter No. 2086 and spindle parameter No. 4127.

- Speedometer

Although the speedometer normally indicates the speed of the spindle motor, it can also be used to indicate the speed of the spindle by setting bit 6 (OPS) of parameter No. 3111 to 1.
The spindle speed to be displayed during operation monitoring is calculated from the speed of the spindle motor (see the formula below).
The spindle speed can therefore be displayed, during operation monitoring, even when no position coder is used.
To display the correct spindle speed, however, the maximum spindle speed for each gear (spindle speed at each gear ratio when the spindle motor rotates at the maximum speed) must be set in parameters Nos. 3741 to 3744.
The input of the clutch and gear signals for the first serial spindle is used to determine the gear which is currently selected. Control the input of the CTH1A and CTH2A signals according to the gear selection, by referring to the table below.
(Formula for calculating the spindle speed to be displayed)

$$\text{Spindle speed displayed during operation monitoring} = \frac{\text{Speed of spindle motor}}{\text{Maximum speed of spindle motor}} \times \text{Maximum spindle speed with the gear being used}$$

The following table lists the correspondence between clutch and gear selection signals CTH1A and CTH2A, used to determine the gear being used, and parameters:

CTH1A	CTH2A	Parameter	Serial spindle specification
0	0	=No.3741 (Maximum spindle speed with gear 1)	HIGH
0	1	=No.3742 (Maximum spindle speed with gear 2)	MEDIUM HIGH
1	0	=No.3743 (Maximum spindle speed with gear 3)	MEDIUM LOW
1	1	=No.3744 (Maximum spindle speed with gear 4)	LOW

The speed of the spindle motor and spindle can be displayed, during operation monitoring, only for the first serial spindle and the spindle switching axis for the first serial spindle. It cannot be displayed for the second spindle.

- Color of graph

If the value of a load meter exceeds 100%, the bar graph turns purple.

12.1.9 Display of 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount)

The absolute coordinates of the tool tip, the number of pulses, and a machine axis move amount based on 3-dimensional manual feed are displayed.

Displaying the screen for 3-dimensional manual feed

Procedure

1 Press function key .
2 Press the continuous menu key  several times to display the soft key [3-D MANUAL].
3 Press the soft key [3-D MANUAL] to display the screen of 3-dimensional manual feed.

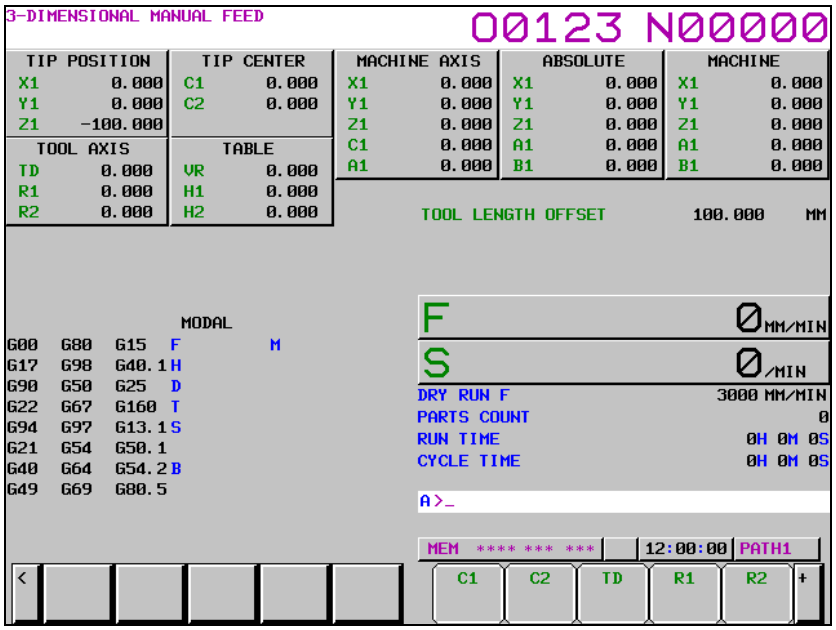


Fig. 12.1.9 (a) 3-dimensional manual feed screen(10.4-inch display unit)

Explanation

- Tool tip position

The addresses of the three basic machine configuration axes for performing 3-dimensional manual feed and the current position of the tool tip are displayed.

- Tool axis reference (number of pulses)

TD

The amount of travel in the tool axis direction in tool axis direction handle feed, tool axis direction jog feed, or tool axis direction incremental feed is displayed.
The unit is the least input increment of the axis in the direction specified by parameter No. 19697.

R1

The amount of travel in the first axis direction in tool axis right-angle direction handle feed, tool axis right-angle direction jog feed, or tool axis right-angle direction incremental feed is displayed.
The unit is the least input increment of the axis in the first axis direction normal to the direction specified by parameter No. 19697.

R2

The amount of travel in the second axis direction in tool axis right-angle direction handle feed, tool axis right-angle direction jog feed, or tool axis normal direction incremental feed is displayed.
The unit is the least input increment of the axis in the second axis direction normal to the direction specified by parameter No. 19697.

- Tool tip center (number of pulses)

C1

The angular displacement in tool tip center rotation handle feed, tool tip center rotation jog feed, or tool tip center rotation incremental feed for rotation of the first rotary axis is displayed. The unit is the least input increment of the first rotary axis.

C2

The angular displacement in tool tip center rotation handle feed, tool tip center rotation jog feed, or tool tip center rotation incremental feed for rotation of the second rotary axis is displayed. The unit is the least input increment of the second rotary axis.

- Table reference (number of pulses)

VR

The amount of travel in the table reference vertical direction in table reference vertical direction handle feed, table reference vertical direction jog feed, or table reference vertical direction incremental feed is displayed.
The unit is the least input increment of the axis in the direction specified by parameter No. 19697.

H1

The amount of travel in the first axis direction in table reference horizontal direction handle feed, table reference horizontal direction jog feed, or table reference horizontal direction incremental feed is displayed.
The unit is the least input increment of the axis in the first axis direction normal to the direction specified by parameter No. 19697.

H2

The amount of travel in the second axis direction in table reference horizontal direction handle feed, table reference horizontal direction jog feed, or table reference horizontal direction incremental feed is displayed.
The unit is the least input increment of the axis in the second axis direction normal to the direction specified by parameter No. 19697.

- Amounts of machine axis travel

The addresses of machine configuration axes used for 3-dimensional manual feed and the sum of the amount of travel of each axis used for 3-dimensional manual feed are displayed.
The values of the basic three axes (X-axis, Y-axis, and Z-axis), the first rotary axis, and the second rotary axis are displayed in this order.
For the definition of the first rotary axis and second rotary axis, see the description of parameter No. 19680.
When bit 0 (CLR) of parameter No. 13113 is set to 1, the displayed data is cleared by a reset.

- Absolute coordinates, machine coordinates

The absolute coordinates and machine coordinates of all axes are displayed. If too many axes are involved for display on one screen, the remaining axes can be displayed by pressing the soft key [3-D MANUAL] for page feed.

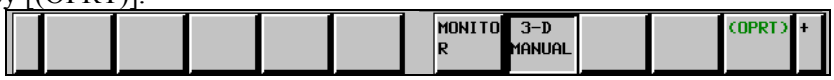
- F (feedrate)

- When bit 3 (CFD) of parameter No. 13113 is set to 0
  The composite feedrate at a control point on a linear axis or rotary axis is displayed.
- When bit 3 (CFD) of parameter No. 13113 is set to 1
  The feedrate of the tool tip is displayed.

Operation

The display of the number of pulses can be cleared by soft key operations.

1 Press soft key [(OPRT)].

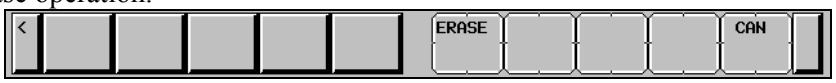


2 Select the soft key corresponding to a function subject to clearing of the amount of travel.
Pressing the rightmost soft key displays the second page.





3 Press soft key [ERASE] to clear the amount of travel of the specified function. Press soft key [CAN] to cancel erase operation.



Screens of a 15/19-inch display unit

Press function key POS to display the current overall position display screen that shows the current position of the tool.
The above screen can also display the feedrate, run time, and the number of parts. In addition, a floating reference position can be set on this screen.
Function key POS can also be used to display the load on the servo motor and spindle motor and the rotation speed of the spindle motor (operating monitor display).
Function key POS can also be used to display the screen for displaying the distance moved by handle interruption. See III-4.6 for details on this screen.

12.1.10 Overall Position Display (15/19-inch Display Unit)

Displays the following positions on a screen : Current positions of the tool in the workpiece coordinate system, relative coordinate system, and machine coordinate system, and the remaining distance.
The relative coordinates can also be set on this screen. (Refer to Explanation.)

Procedure for displaying overall position display screen

Procedure

1 Press function key POS.
2 Press vertical soft key [ALL].

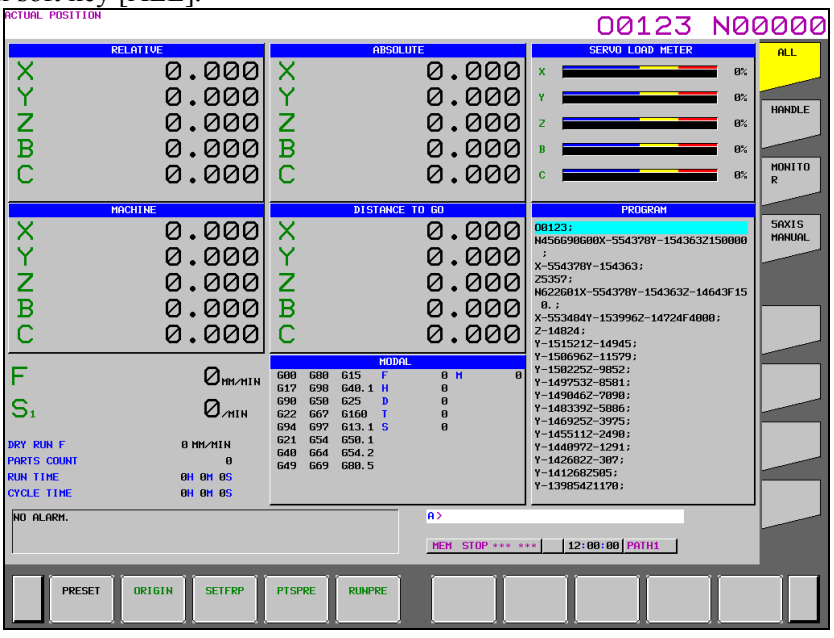


Fig. 12.1.10 (a) Current position screen (M series) (15-inch display unit)

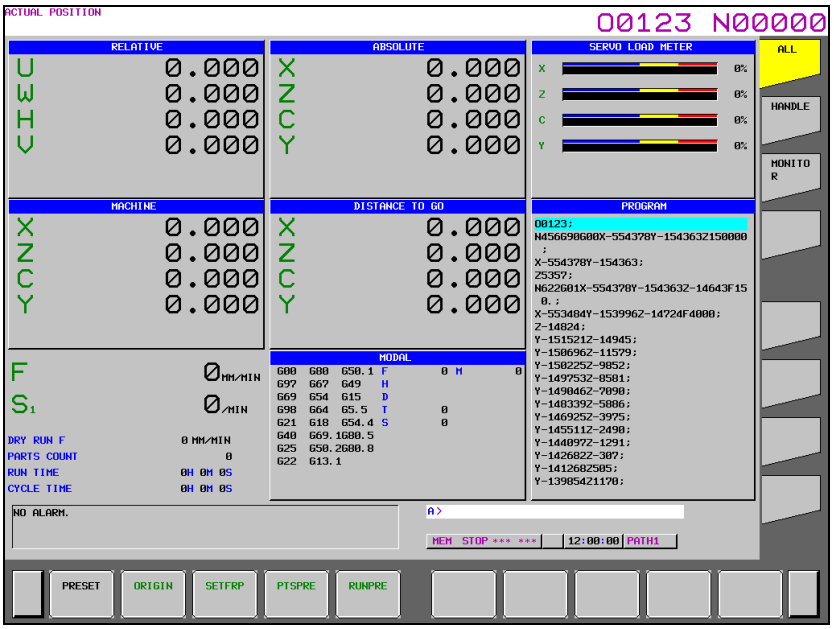


Fig. 12.1.10 (b) Current position screen (T series) (15-inch display unit)

Explanation

- Coordinate display

The current positions of the tool in the following coordinate systems are displayed at the same time:

- Current position in the relative coordinate system (relative coordinate)
- Current position in the work coordinate system (absolute coordinate)
- Current position in the machine coordinate system (machine coordinate)
- Distance to go (distance to go)

- Distance to go

The distance remaining is displayed in the MEM or MDI mode.
The distance the tool is yet to be moved in the current block is displayed.

- Machine coordinate system

The least command increment is used as the unit for values displayed in the machine coordinate system. However, the least input increment can be used by setting bit 0 (MCN) of parameter No. 3104.

- Setting the relative coordinates

The current position of the tool in the relative coordinate system can be reset to 0 or preset to a specified value as follows.
Normal decimal point input and Pocket calculator type decimal point input can be selected by bit 0 (DPI) of parameter.No.3401.

Resetting relative coordinates to 0

When all axes are reset to 0

1 Press function key 

.



2 Press horizontal soft key [ORIGIN].



3 Press horizontal soft key [ALL AXIS].
The relative coordinates of all axes are reset to 0.

When a specified axis is reset to 0

1 Press function key [POS].



2 Press horizontal soft key [ORIGIN].



3 Input the axis name of the resetting axis.
The axis name blinks.
4 Press horizontal soft key [EXEC].
The relative coordinate is reset to 0.

Presetting relative coordinates

Procedure

1 Press function key [POS].



2 Input the axis name of the presetting axis.
The axis name blinks.
3 Input the coordinates value, and press horizontal soft key [PRESET].
The relative coordinate is preset by input coordinate value.

NOTE
1 When the preset of the specified axis is done in the following states, warning is displayed.
- When only the numerical value is input or the character that is not the axis name is input
  Waning massage: “THERE IS NO SPECIFIED NAME OF AXIS”
- When input form is except for “axis name + numerical value” is done
  Waning massage: “FORMAT ERROR”

2 When axis name display is changed by parameter No.3132, specify the changed axis name.

- Display including compensation values

M

The following parameters can be used to determine whether the displayed values include tool length compensation and cutter compensation.

- Workpiece coordinate system: Bits 6 (DAL) and 7 (DAC) of parameter No. 3104
- Relative coordinate system: Bits 4 (DRL) and 5 (DRC) of parameter No. 3104

T

The following parameters can be used to determine whether the displayed values include tool offset and tool nose radius compensation.

- Workpiece coordinate system: Bit 1 (DAP) of parameter No. 3129 and bit 7 (DAC) of parameter No. 3104
- Relative coordinate system: Bit 0 (DRP) of parameter No. 3129 and bit 5 (DRC) of parameter No. 3104

- Presetting by setting a coordinate system

M

Bit 3 (PPD) of parameter No. 3104 can be used to specify whether the position indication values in the absolute coordinate system are preset as those in the relative coordinate system during coordinate system setting or manual reference position return by the G92 command.

T

Bit 3 (PPD) of parameter No. 3104 can be used to specify whether the position indication values in the absolute coordinate system are preset as those in the relative coordinate system during coordinate system setting or manual reference position return by the G50 command (for G code system A) or G92 command (for G code system B or C).

12.1.11 Workpiece Coordinate System Preset (15/19-inch Display Unit)

If a workpiece coordinate system has been shifted with manual intervention or any other operation, an MDI operation can be performed to preset the system to a workpiece coordinate system that is offset by a workpiece origin offset, from the machine zero point that has been set before the shifting.
A command (G92.1) can be programmed to preset a workpiece coordinate system. (See II-7.2.4.)

Procedure for the workpiece coordinate system preset

Procedure

1 Press function key [POS].
2 Press vertical soft key [ALL].
3 Enter the axis name ([X], [&Y], ...) and zero ([P 0]).
4 Press soft key [PRESET].

Explanation

- Operation mode

This function can be executed when the reset state or automatic operation stop state is entered, regardless of the operation mode.

- Presetting relative coordinates

As with absolute coordinates, bit 3 (PPD) of parameter No. 3104 is used to specify whether to preset relative coordinates (RELATIVE).

12.1.12 Actual Feedrate Display (15/19-inch Display Unit)

The actual feedrate on the machine (per minute) can be displayed on a current position display screen or program check screen.

Display procedure for the actual feedrate on the current position display screen

Procedure

1 Press the function key [POS] to display a current position display screen. At the location indicated by [ ], an actual feedrate is displayed.

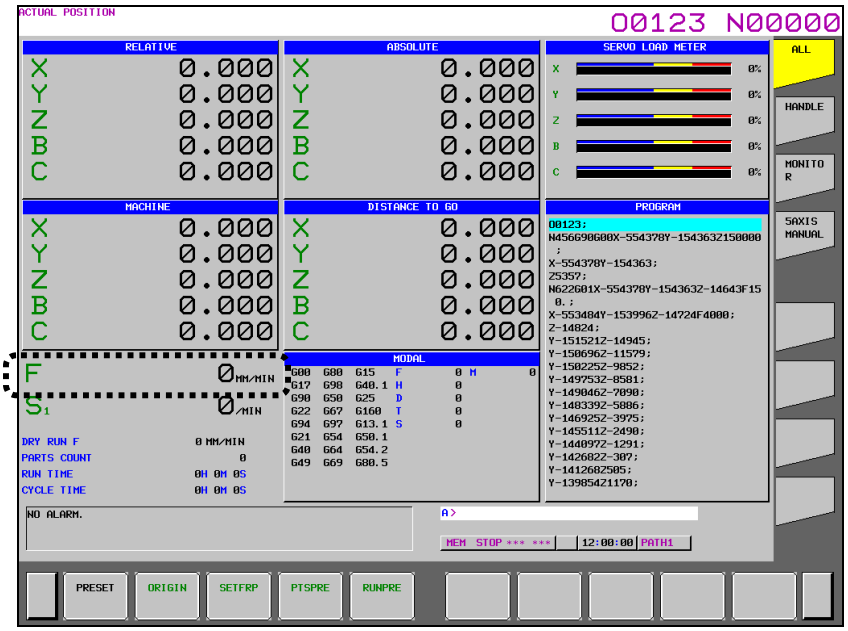


Fig. 12.1.12 (a) Current position screen (M series) (15-inch display unit)

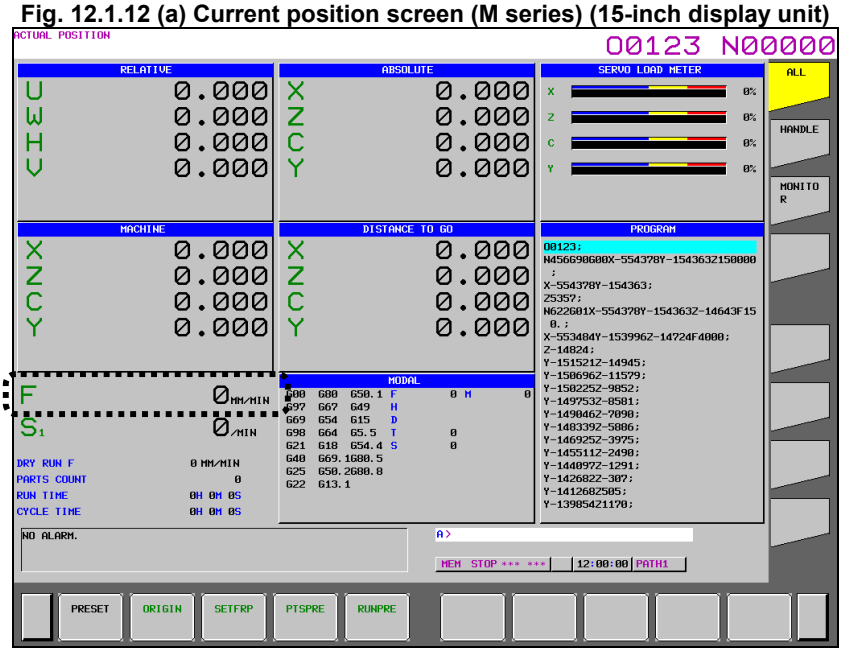


Fig. 12.1.12 (b) Current position screen (T series) (15-inch display unit)

The actual feedrate is displayed in units of millimeter/min or inch/min (depending on the specified least input increment) under the display of the current position.

Explanation

- Actual feedrate value

The actual rate is calculated by the following expression:

$$Fact = \sqrt{\sum_{i=1}^{n} (fi)^2}$$

n : Number of axes
fi : Cutting feed rate in the tangential direction of each axis or rapid traverse rate
Fact : Actual feedrate displayed

The display unit:

mm/min (metric input).
inch/min (Inch input, two digits below the decimal point are displayed.)

- Actual feedrate display of feed per revolution

In the case of feed per revolution and thread cutting, the actual feedrate displayed is the feed per minute rather than feed per revolution.

- Actual feedrate display of rotary axis

In the case of movement of rotary axis, the speed is displayed in units of deg/min but is displayed on the screen in units of input system at that time. For example, when the rotary axis moves at 50 deg/min, the following is displayed: 50 mm/min (in metric input) or 0.50 inch/min (in inch input).

- Actual feedrate display on the other screen

The program check screen also displays the actual feedrate.

12.1.13 Display of Run Time and Parts Count (15/19-inch Display Unit)

The run time, cycle time, and the number of machined parts are displayed on the current position display screens.

Procedure for displaying run time and parts count on the current position display screen

Procedure

1 Press the function key 

to display a current position display screen. At the location indicated by ⬚, a run time and parts count are displayed.

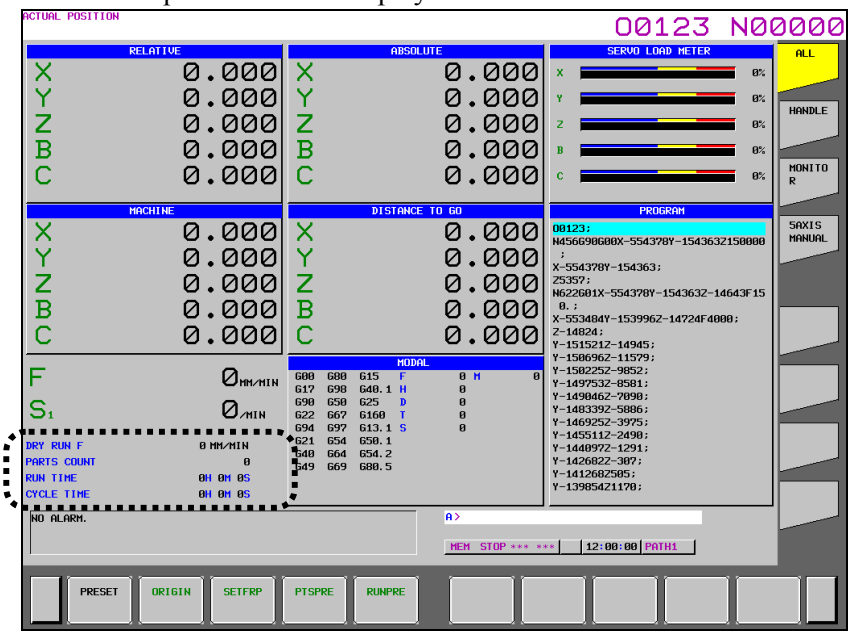


Fig. 12.1.13 (a) Current position screen (M series) (15-inch display unit)

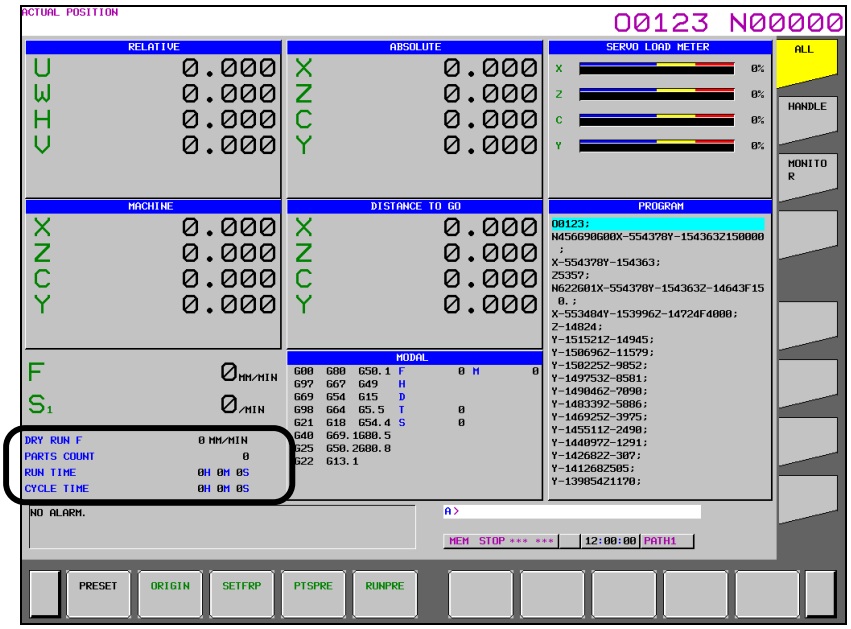


Fig. 12.1.13 (b) Current position screen (T series) (15-inch display unit)

The number of machined parts (PART COUNT), run time (RUN TIME), and cycle time (CYCLE TIME) are displayed under the current position.

Explanation

- PART COUNT

Indicates the number of machined parts. The number is incremented each time M02, M30, or an M code specified by parameter No. 6710 is executed.

- RUN TIME

Indicates the total run time during automatic operation, excluding the stop and feed hold time.

- CYCLE TIME

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- Display on the other screen

Details of the run time and the number of machined parts are displayed on the setting screen. See III-12.3.20.

- Parameter setting

The number of machined parts and run time cannot be set on current position display screens.
The number of machined parts and run time cannot be set on current position display screens. They can be set by parameters Nos. 6711, 6751, and 6752 or on the setting screen.

- Incrementing the number of machined parts

Bit 0 (PCM) of parameter No. 6700 is used to specify whether the number of machined parts is incremented each time M02, M30, or an M code specified by parameter No. 6710 is executed, or only each time an M code specified by parameter No. 6710 is executed.

12.1.14 Setting the Floating Reference Position (15/19-inch Display Unit)

To perform floating reference position return with a G30.1 command, the floating reference position must be set beforehand.

Procedure for setting the floating reference position

Procedure

1 Press function key  to display a screen used for displaying the current position.
2 Move the tool to the floating reference position by jogging.
3 Press horizontal soft key [SET FRP].
4 To register the floating reference positions for all axes, press soft key [ALL AXIS].
To register the floating reference position of a specific axis, enter the name of the axis (, , etc.), then press horizontal soft key [AXIS NAME]. Two or more names can be entered consecutively (e.g.,)   [AXIS NAME]).
The above operation stores the floating reference position. It can be checked with parameter No. 1244.
5 In step 3, the floating reference position along a specified axis can also be stored by entering the axis name (such as  or ) and pressing horizontal soft key [SET FRP].

Explanation

- Presetting the relative coordinate system

When bit 3 (FPC) of parameter No. 1201 is set to 1, the floating reference position can be registered, while the relative position display can be preset to 0.

12.1.15 Operating Monitor Display (15/19-inch Display Unit)

The load meter for each servo axis can be displayed. Also, the load meter and speed meter for a serial spindle can be displayed.
To enable this function, bit 5 (OPM) of parameter No. 3111 must be set to 1.

Procedure for displaying the operating monitor

Procedure

1 Press function key .
2 Press vertical soft key [MONITOR].

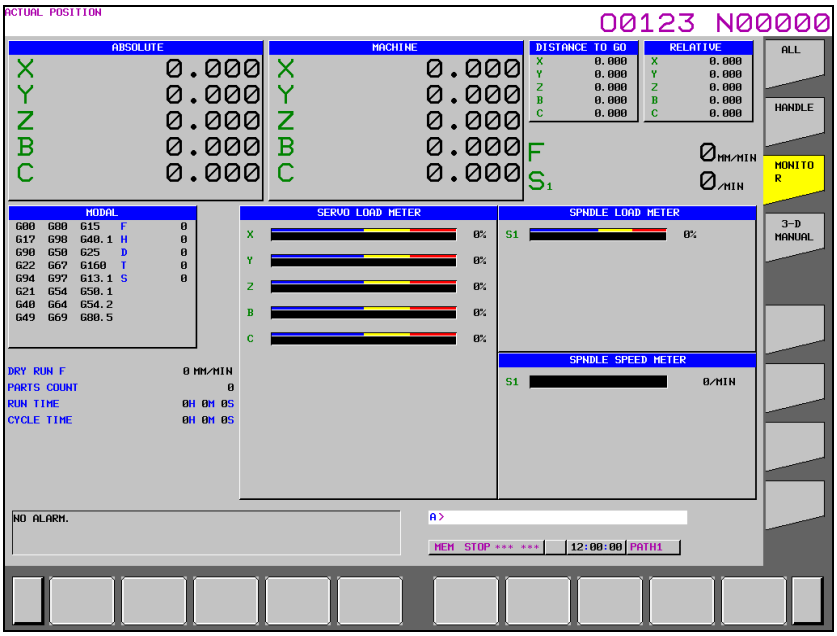


Fig. 12.1.15 (a) Operating monitor (M series) (15-inch display unit)

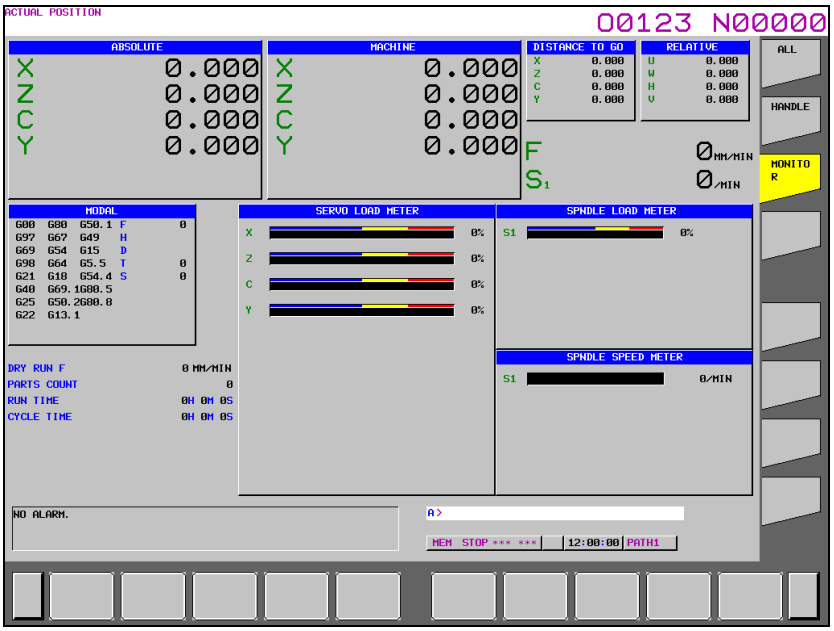


Fig. 12.1.15 (b) Operating monitor (T series) (15-inch display unit)

Explanation

- Display of the servo axes

Servo axis load meters as many as the maximum number of controlled axes of the path can be displayed. One screen displays load meters for up to five axes at a time. By pressing the vertical soft key [MONITOR], load meters for the 6th axis and up are displayed.

- Display of the spindle axes

When serial spindles are used, the reading on the load meter and speedometer can be displayed only for the main serial spindle.

- Unit of graph

The bar graph for the load meter shows load up to 200% (only a value is displayed for load exceeding 200%).

The bar graph for the speedometer shows the ratio of the current spindle speed to the maximum spindle speed (100%).

- Load meter

The reading on the load meter depends on servo parameter No. 2086 and spindle parameter No. 4127.

- Speedometer

Although the speedometer normally indicates the speed of the spindle motor, it can also be used to indicate the speed of the spindle by setting bit 6 (OPS) of parameter No. 3111 to 1.
The spindle speed to be displayed during operation monitoring is calculated from the speed of the spindle motor (see the formula below).
The spindle speed can therefore be displayed, during operation monitoring, even when no position coder is used.
To display the correct spindle speed, however, the maximum spindle speed for each gear (spindle speed at each gear ratio when the spindle motor rotates at the maximum speed) must be set in parameters Nos. 3741 to 3744.
The input of the clutch and gear signals for the first serial spindle is used to determine the gear which is currently selected. Control the input of the CTH1A and CTH2A signals according to the gear selection, by referring to the table below.
(Formula for calculating the spindle speed to be displayed)

$$\text{Spindle speed displayed during operation monitoring} = \frac{\text{Speed of spindle motor}}{\text{Maximum speed of spindle motor}} \times \text{Maximum spindle speed with the gear being used}$$

The following table lists the correspondence between clutch and gear selection signals CTH1A and CTH2A, used to determine the gear being used, and parameters:

CTH1A	CTH2A	Parameter	Serial spindle specification
0	0	=No.3741 (Maximum spindle speed with gear 1)	HIGH
0	1	=No.3742 (Maximum spindle speed with gear 2)	MEDIUM HIGH
1	0	=No.3743 (Maximum spindle speed with gear 3)	MEDIUM LOW
1	1	=No.3744 (Maximum spindle speed with gear 4)	LOW

The speed of the spindle motor and spindle can be displayed, during operation monitoring, only for the first serial spindle and the spindle switching axis for the first serial spindle. It cannot be displayed for the second spindle.

- Color of graph

If the value of a load meter exceeds 100%, the bar graph turns red.

12.1.16 Display of 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount) (15/19-inch Display Unit)

The absolute coordinates of the tool tip, the number of pulses, and a machine axis move amount based on 3-dimensional manual feed are displayed.

Displaying the screen for 3-dimensional manual feed

Procedure

1 Press function key 

.

2 Press the vertical soft key [3-D MANUAL] to display the screen of 3-dimensional manual feed.

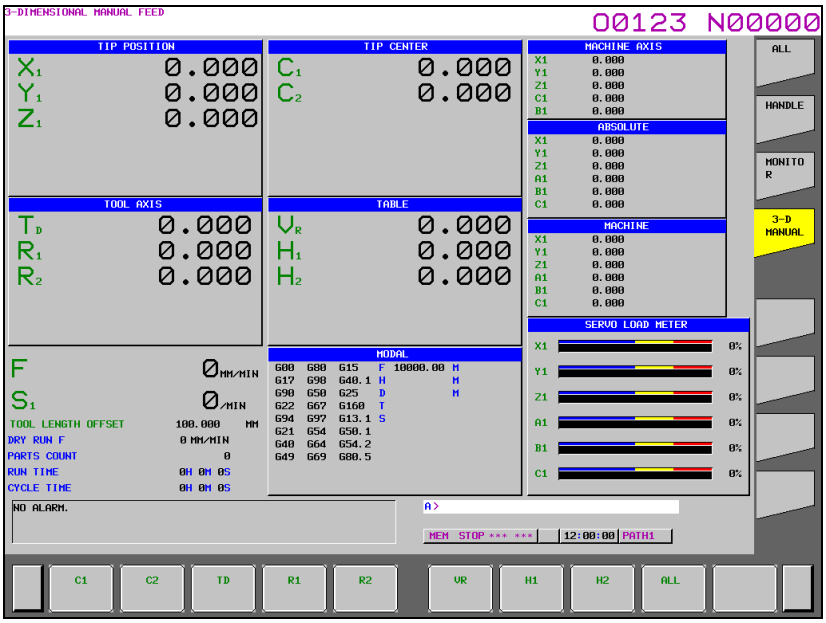


Fig. 12.1.16 (a) 3-dimensional manual feed screen(15-inch display unit)

Explanation

- Tool tip position

The addresses of the three basic machine configuration axes for performing 3-dimensional manual feed and the current position of the tool tip are displayed.

- Tool axis reference (number of pulses)

TD

The amount of travel in the tool axis direction in tool axis direction handle feed, tool axis direction jog feed, or tool axis direction incremental feed is displayed.
The unit is the least input increment of the axis in the direction specified by parameter No. 19697.

R1

The amount of travel in the first axis direction in tool axis right-angle direction handle feed, tool axis right-angle direction jog feed, or tool axis right-angle direction incremental feed is displayed.
The unit is the least input increment of the axis in the first axis direction normal to the direction specified by parameter No. 19697.

R2

The amount of travel in the second axis direction in tool axis right-angle direction handle feed, tool axis right-angle direction jog feed, or tool axis normal direction incremental feed is displayed.
The unit is the least input increment of the axis in the second axis direction normal to the direction specified by parameter No. 19697.

- Tool tip center (number of pulses)

C1

The angular displacement in tool tip center rotation handle feed, tool tip center rotation jog feed, or tool tip center rotation incremental feed for rotation of the first rotary axis is displayed. The unit is the least input increment of the first rotary axis. For the definition of the first rotary axis, see the description of parameter No. 19680.

C2

The angular displacement in tool tip center rotation handle feed, tool tip center rotation jog feed, or tool tip center rotation incremental feed for rotation of the second rotary axis is displayed. The unit is the least input increment of the second rotary axis. For the definition of the second rotary axis, see the description of parameter No. 19680.

- Table reference (number of pulses)

VR

The amount of travel in the table reference vertical direction in table reference vertical direction handle feed, table reference vertical direction jog feed, or table reference vertical direction incremental feed is displayed.
The unit is the least input increment of the axis in the direction specified by parameter No. 19697.

H1

The amount of travel in the first axis direction in table reference horizontal direction handle feed, table reference horizontal direction jog feed, or table reference horizontal direction incremental feed is displayed.
The unit is the least input increment of the axis in the first axis direction normal to the direction specified by parameter No. 19697.

H2

The amount of travel in the second axis direction in table reference horizontal direction handle feed, table reference horizontal direction jog feed, or table reference horizontal direction incremental feed is displayed.
The unit is the least input increment of the axis in the second axis direction normal to the direction specified by parameter No. 19697.

- Amounts of machine axis travel

The addresses of machine configuration axes used for 3-dimensional manual feed and the sum of the amount of travel of each axis used for 3-dimensional manual feed are displayed.
The values of the basic three axes (X-axis, Y-axis, and Z-axis), the first rotary axis, and the second rotary axis are displayed in this order.
For the definition of the first rotary axis and second rotary axis, see the description of parameter No. 19680.
When bit 0 (CLR) of parameter No. 13113 is set to 1, the displayed data is cleared by a reset.

- Absolute coordinates, machine coordinates

The absolute coordinates and machine coordinates of all axes are displayed. If too many axes are involved for display on one screen, the remaining axes can be displayed by pressing the vertical soft key [3-D MANUAL] for page feed.

- F (feedrate)

- When bit 3 (CFD) of parameter No. 13113 is set to 0
  The composite feedrate at a control point on a linear axis or rotary axis is displayed.
- When bit 3 (CFD) of parameter No. 13113 is set to 1
  The feedrate of the tool tip is displayed.

Operation

The display of the number of pulses can be cleared to 0 with horizontal soft keys.

1 Select the horizontal soft key corresponding to a function for which the display of the amount of travel is to be cleared.



2 Press horizontal soft key [ERASE] to clear the display of the amount of travel of the specified function, or press horizontal soft key [CAN] to cancel the operation.



12.2 SCREENS DISPLAYED BY FUNCTION KEY

Section 12.2, "SCREENS DISPLAYED BY FUNCTION KEY ", consists of the following subsections:

——————— Screens of a 8.4/10.4-inch display unit
12.2.1 Program Contents Display....1372
12.2.2 Editing a Program....1376
12.2.3 Program Screen for MDI Operation....1377
12.2.4 Program Folder Screen....1378
12.2.5 Next Block Display Screen....1385
12.2.6 Program Check Screen....1386
12.2.7 Background Editing....1386
12.2.8 Stamping the Machining Time....1392
12.2.9 Screen for Assistance in Entering Tilted Working Plane Indexing....1399

——————— Screens of a 15/19-inch display unit
12.2.10 Program Contents Display (15/19-inch Display Unit)....1416
12.2.11 Editing a Program (15/19-inch Display Unit)....1417
12.2.12 Program Screen for MDI Operation (15/19-inch Display Unit)....1419
12.2.13 Program Folder Screen (15/19-inch Display Unit)....1419
12.2.14 Next Block Display Screen (15/19-inch Display Unit)....1424
12.2.15 Program Check Screen (15/19-inch Display Unit)....1425
12.2.16 Background Editing (15/19-inch Display Unit)....1426
12.2.17 Stamping the Machining Time (15/19-inch Display Unit)....1431
12.2.18 Screen for Assistance in Entering Tilted Working Plane Indexing (15/19-inch Display Unit) 1439

Screens of a 8.4/10.4-inch display unit

This section describes the screens displayed by pressing function key . The screens include a program editing screen, program folder list display screen, and screens for displaying the command states of the program currently being executed.

1. Program screen
2. Program folder screen
3. Next block display screen
4. Program check screen

On the program screen, you edit the program that is currently selected, and display the block that is currently executed during program operation. In MDI mode, you also edit an MDI operation program, and display the block that is currently executed.

12.2.1 Program Contents Display

Displays the program currently being executed in MEM mode.

Displaying the program being executed

Procedure

1 Press function key to display the program screen.

2 Press chapter selection soft key [PROGRAM].
The cursor is positioned at the block currently being executed.

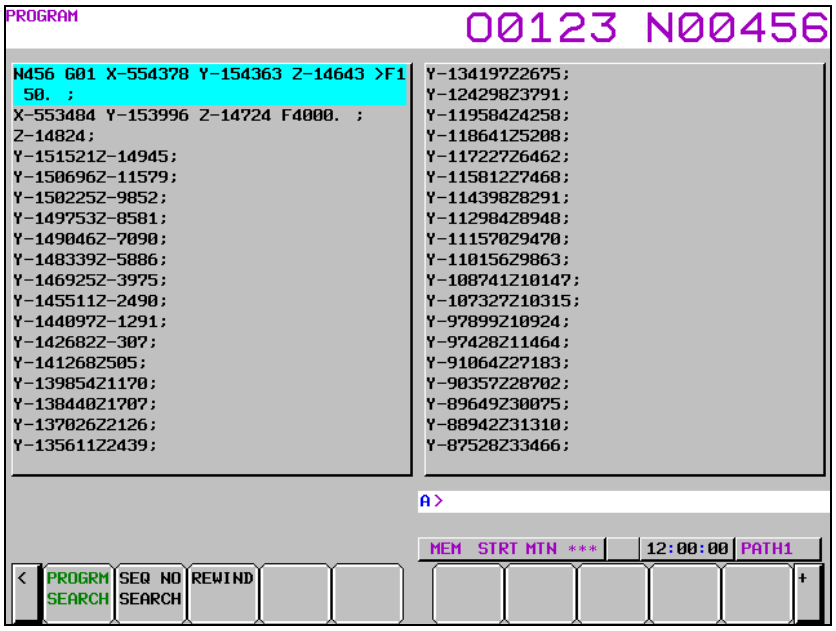


Fig. 12.2.1 (a) Screen for displaying the program being executed (10.4-inch display unit)

12.2.1.1 Displaying the executed block

Overview

When the program being executed is displayed, one executed block can be displayed. This function can be enabled by setting bit 3 (FPD) of parameter No. 11308.

The program screen shows the following:

- Executed block
- Block being executed
- Look-ahead block(*)
- Non-look-ahead block

(*) For details on look-ahead, see the explanation "Program look-ahead" below.

The executed block is displayed on the top of the screen, and ">" is shown in front of the block being executed, across which the cursor is placed. Below the block being executed, look-ahead blocks are displayed. Below them, non-look-ahead blocks are displayed.
There is a dividing line between the look-ahead blocks and the non-look-ahead blocks.

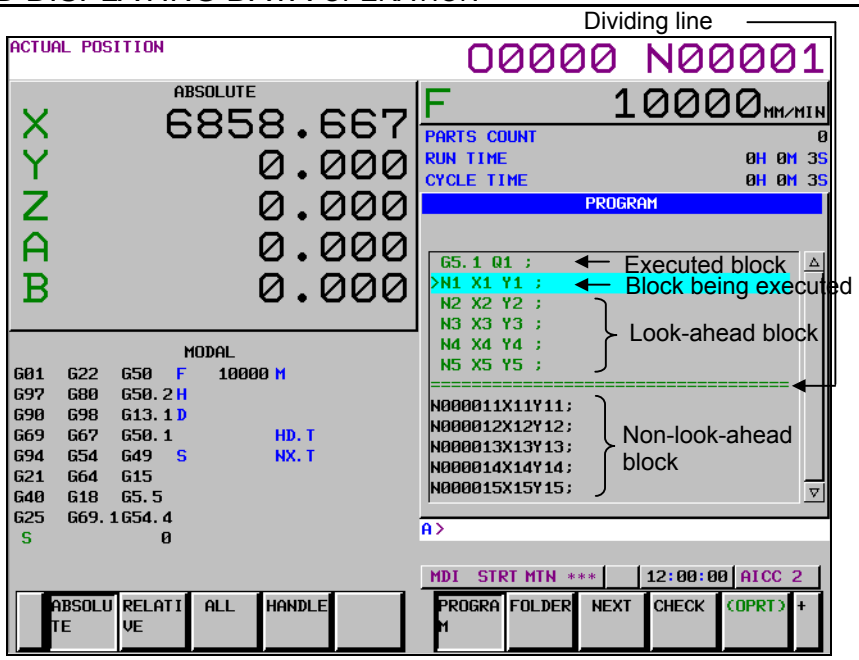


Fig. 12.2.1.1 (b) Screen for displaying the program being executed (MDI mode)

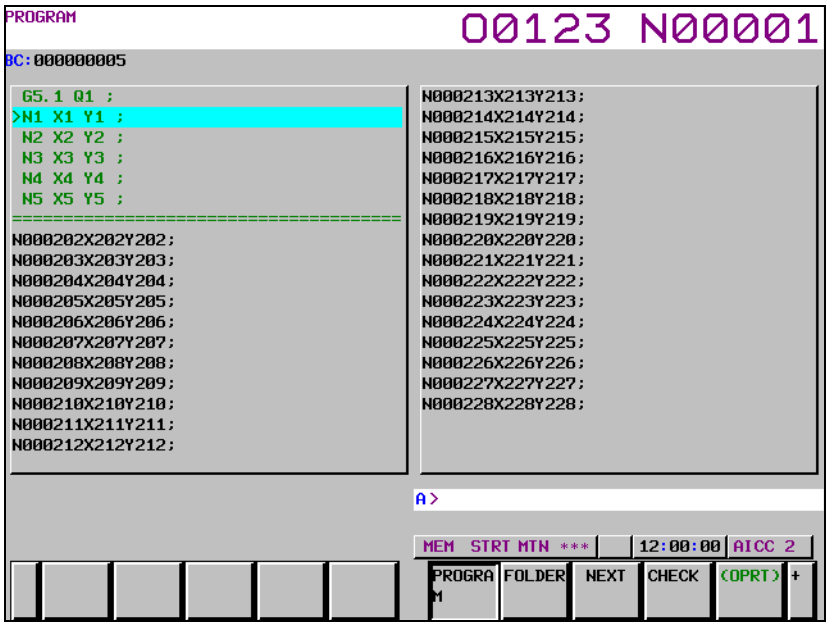


Fig. 12.2.1.1 (c) Screen for displaying the program being executed (MEM mode)

Explanation

- Program look-ahead

When an automatic operation cycle starts, a program is executed as described below.

<1> The command of one block is read from the specified program.
<2> The read command is analyzed and converted to executable data.
<3> The command of the block is executed.

During execution of the command, steps <1> and <2> are performed for the next block in advance. Upon completion of the current block, execution of the next block can be started immediately. This processing is referred to as look-ahead or buffering.

The number of look-ahead blocks depends on the processing status or parameters.

12.2.1.2 Text display

Overview

Setting bit 1 (APD) of parameter No. 11350 can select whether to display the contents of the running NC program in the look-ahead or text mode.

In the look-ahead mode (APD = 0):

Displays look-ahead blocks and non-look-ahead blocks in the running NC program.

In the text mode (APD = 1):

Displays blocks in the running NC program that is in the program memory. The displayed blocks include the block just before the block being executed.

(Example)

Fig. 12.2.1.2 (d) shows the display in the look-ahead mode and Fig. 12.2.1.2 (e) shows the display in the text mode when block N4 in program O0099 shown below is being executed.

```
O0099
N1 G91 G00 X100.
N2 Z100.
N3 #501 = 100.
N4 Y-100.
IF[#1 GT 10] GOTO 7
N5 Y#501 (COMMENT)
N6 M98 P9100
N7 G90 X100
N8 G300
N9 G00 X0Y0
N10 M30
```

Running program

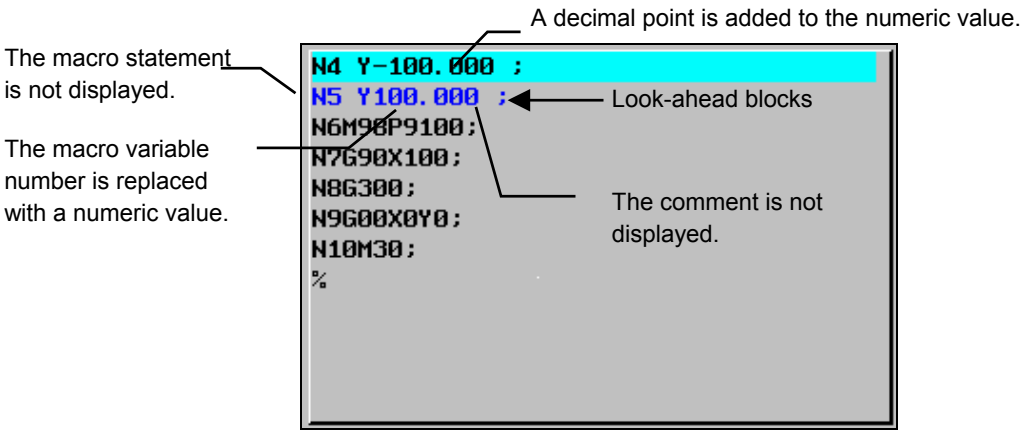


Fig. 12.2.1.2 (d) Display when block N4 is being executed (look-ahead mode)

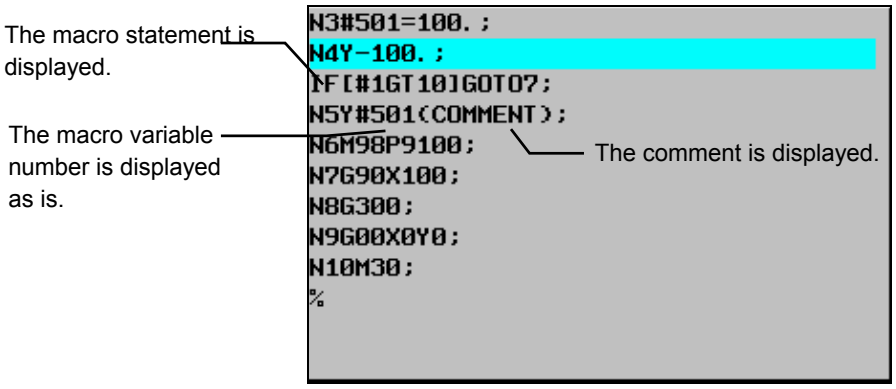


Fig. 12.2.1.2 (e) Display when block N4 is being executed (text mode)

External subprogram call

Setting bit 7 (DPD) of parameter No. 11356 can select whether to display the contents of the running external subprogram by DNC operation or M198 in the look-ahead or text mode.
However, text mode is effective when the number of look-ahead blocks is only less than 5.
When the number of look-ahead blocks is 5 or more, look-ahead mode is effective.

12.2.2 Editing a Program

A program can be edited in the EDIT mode.
Two modes of editing are available. One mode is word editing, which performs word-by-word editing. The other is character editing, which performs character-by-character editing.
For program creation and editing operation, see Chapter III-9, “CREATING PROGRAMS” and Chapter III-10, “EDITING PROGRAMS”.

Displaying the program editing screen

Procedure

1 Press function key 

to display the program screen.

2 Press chapter selection soft key [PROGRAM].

- Word editing

Editing operations such as text insertion, modification, and deletion, and cursor movements are performed on a word-by-word basis.

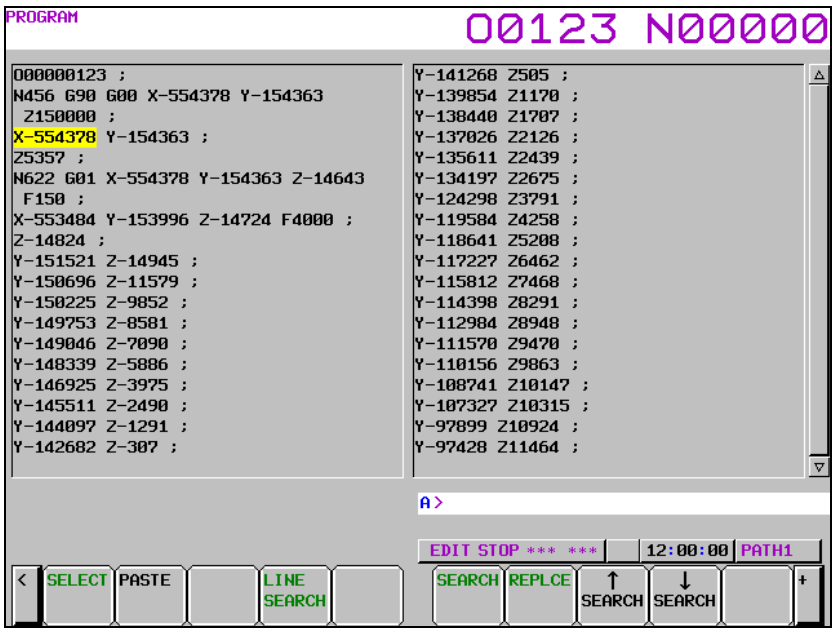


Fig. 12.2.2 (a) Program word editing screen (10.4-inch display unit)

- Character editing

Program editing operations and cursor movements are performed on a character-by-character basis as with a general text editor.
Text is input directly to the cursor position instead of using the key input buffer.

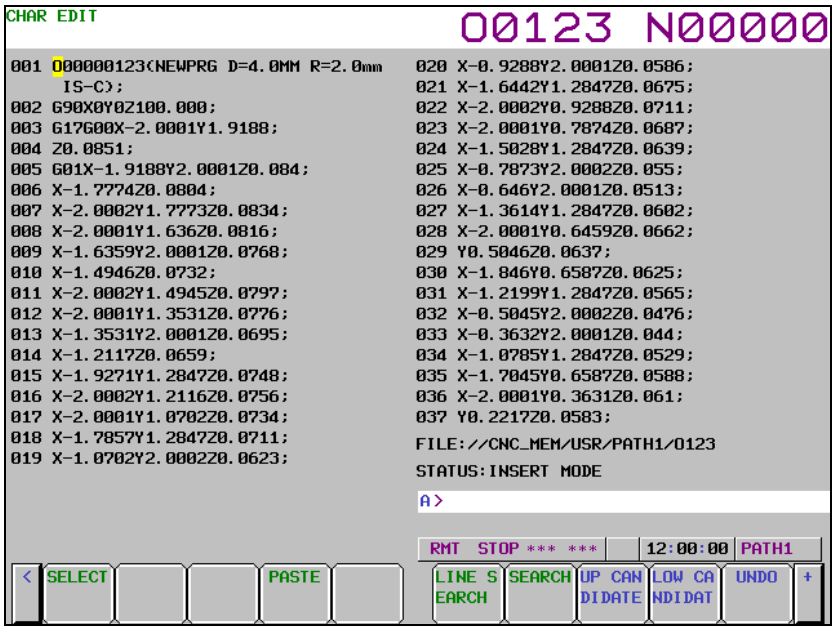


Fig. 12.2.2 (b) Program character editing screen (10.4-inch display unit)

Switching between program editing modes

You can switch between word editing and character editing with soft keys.

Procedure

1 Press function key 

to display the program screen.
2 Press chapter selection soft key [PROGRAM].
3 Press operation soft key [(OPRT)].
4 Pressing operation soft key [CHANGE EDITOR] switches the editing mode between word editing and character editing.

12.2.3 Program Screen for MDI Operation

During MDI operation or editing of an MDI operation program in the MDI mode, the program currently being executed mode is displayed.
For MDI operation, see Section III-4.2, “MDI Operation”.

Procedure for displaying the program screen for MDI operation

Procedure

1 Press function key 

.
2 Press chapter selection soft key [PROGRAM].
The program input from the MDI is displayed.

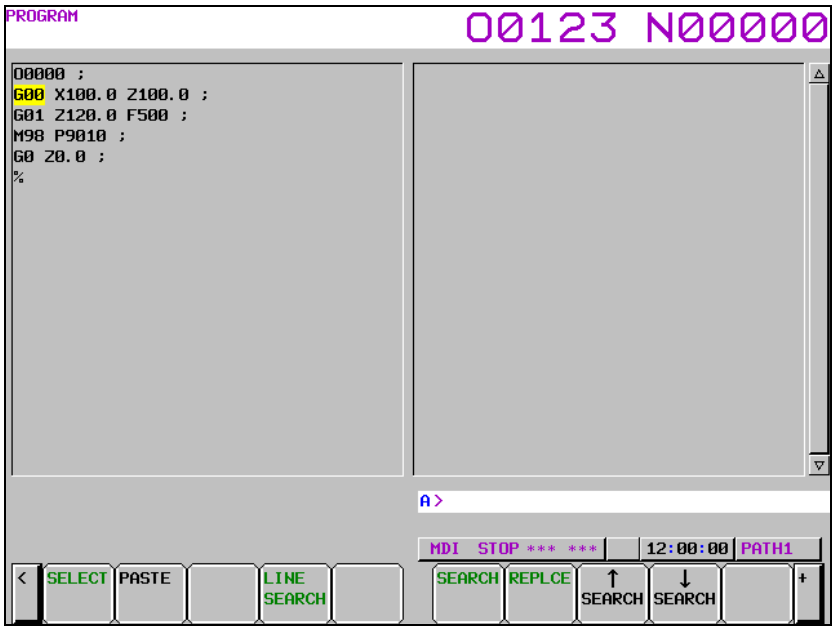


Fig. 12.2.3 (a) MDI operation program screen (10.4-inch display unit)

When bit 7 (MDL) of parameter No. 3107 is 1, modal information is displayed in 8.4-inch display unit.

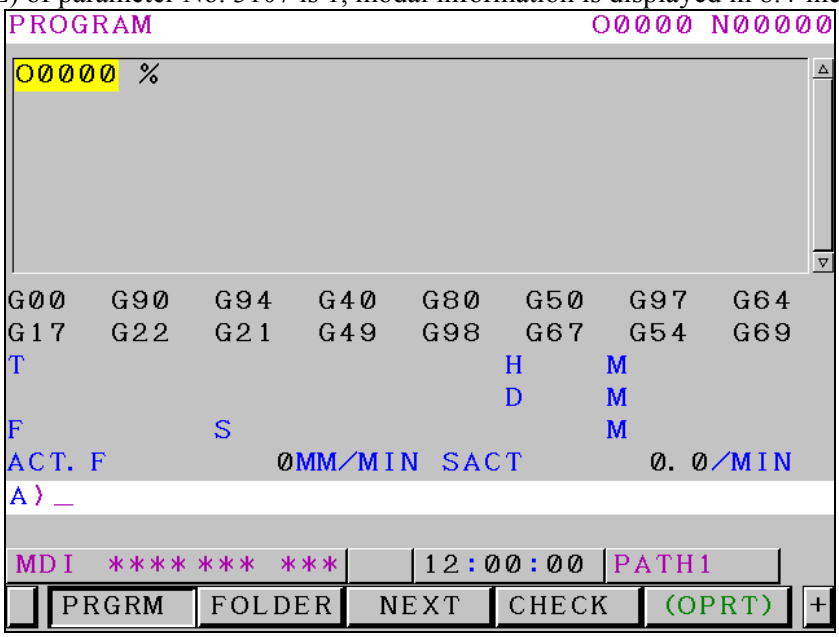


Fig. 12.2.3 (b) Program character editing screen (8.4-inch display unit)

12.2.4 Program Folder Screen

A list of programs registered in the program memory is displayed.
For the program folder screen, see Chapter III-11, “PROGRAM MANAGEMENT”.

Displaying the program folder screen

Procedure

1 Press function key 

.

2 Press chapter selection soft key [FOLDER].

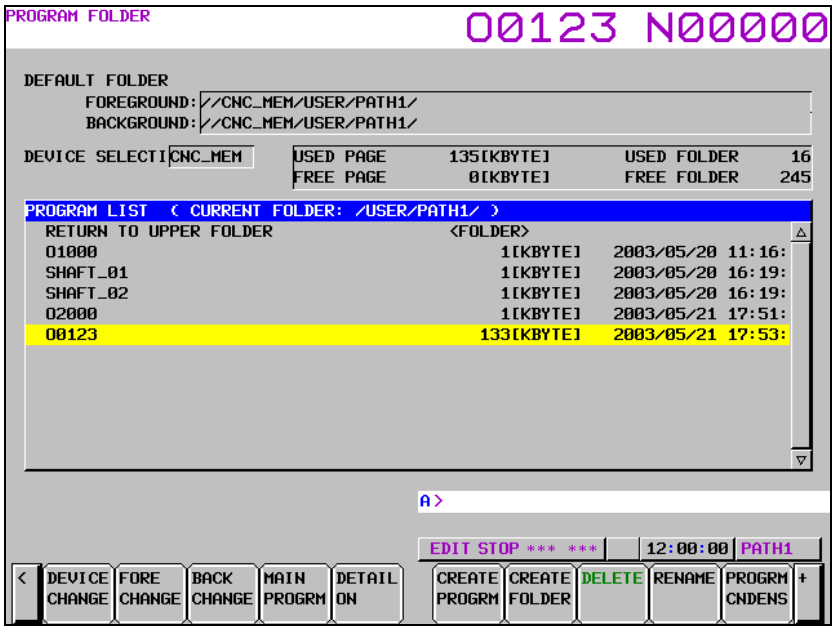


Fig. 12.2.4 (a) Program folder screen (10.4-inch display unit)

12.2.4.1 Split display on the program folder screen

Overview

On the program folder screen, the folder information display can be split into two folder information views, upper and lower, as shown in the Fig. 12.2.4.1 (a). This function is enabled by setting bit 1 (MLD) of parameter No. 11318 to 1.

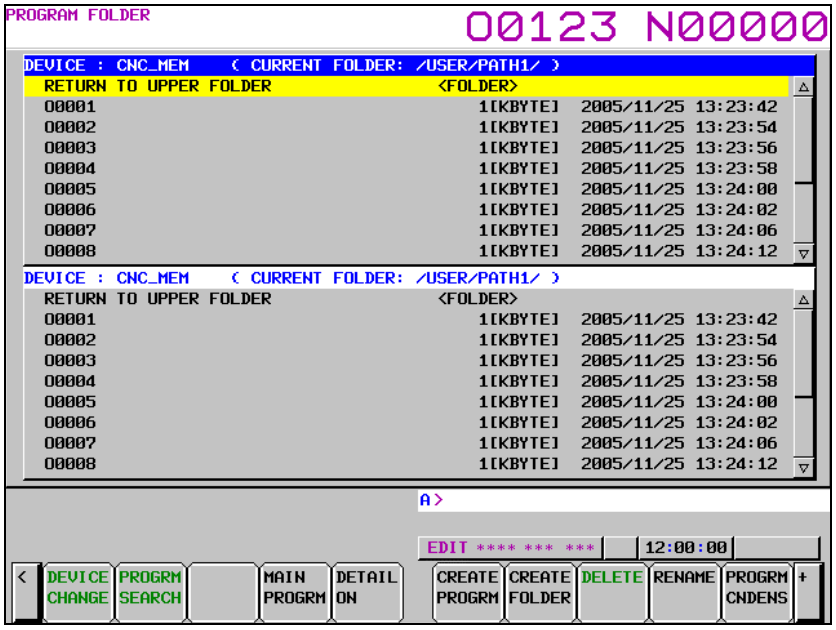
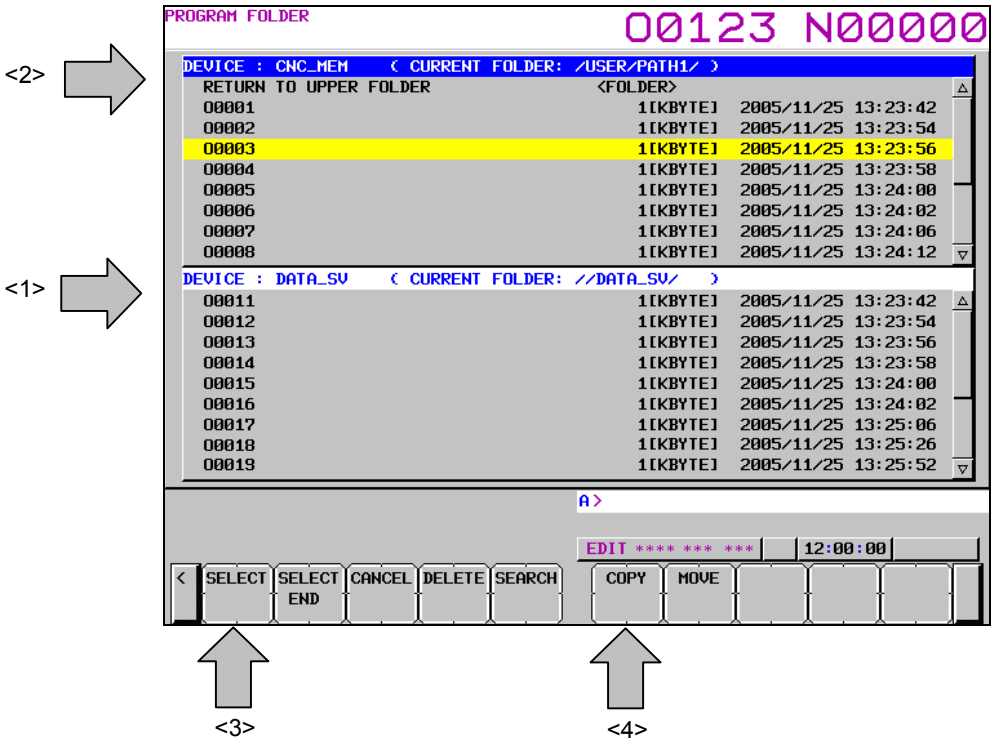


Fig. 12.2.4.1 (a) Program folder screen (split-display screen)(10.4-inch display unit)

On the split folder display, file information about different sets of devices or folders can be shown in different views at the same time, so you can copy or move files easily.

Example of copying a file from the CNC MEM to the Data Server



Procedure

<1> Change to a destination folder on the Data Server.
<2> Change to a folder on the CNC_MEM that contains a file you want to copy.
<3> Select the file.
<4> Copy the file.

Procedure for switching to split display

Procedure

1 Press function key  to display the program folder screen.
2 Press soft key [(OPRT)].
3 Press continuous menu key  twice.
Soft key [MULTI LIST] appears.

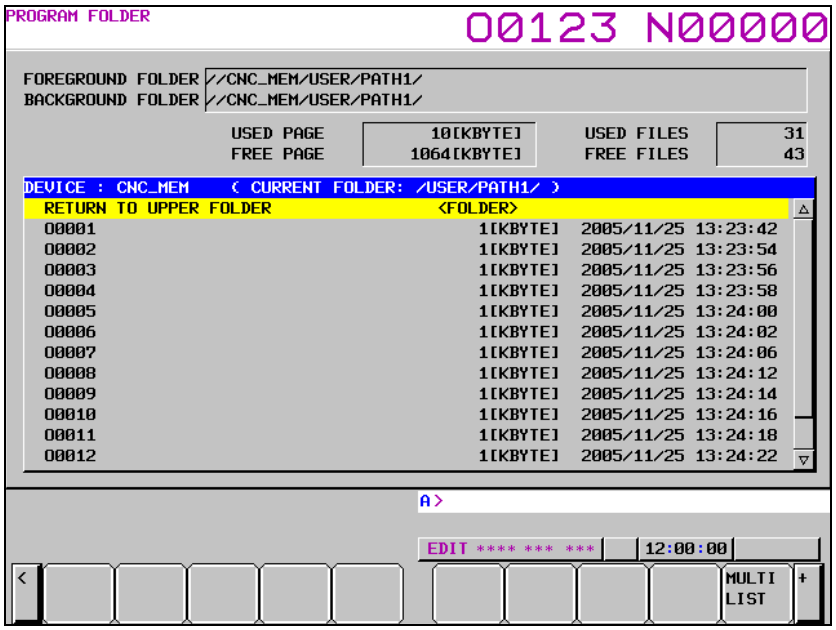


Fig. 12.2.4.1 (b) Program folder screen (normal screen)(10.4-inch display unit)

4 Press soft key [MULTI LIST].
The folder information display is split into two folder views, upper and lower, which show the same folder information. Immediately after the splitting, the upper folder view becomes active for operations. At the same time, soft key [SINGLE LIST] appears.

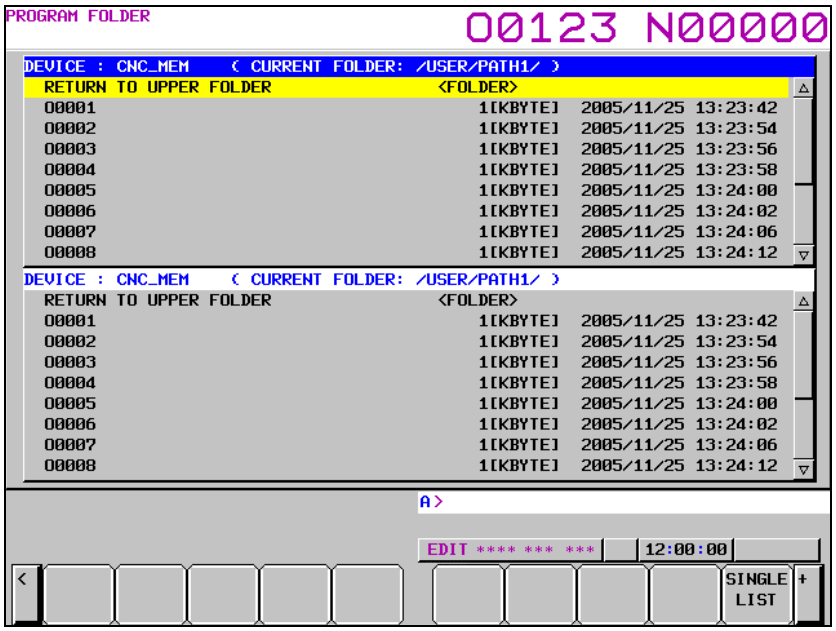


Fig. 12.2.4.1 (c) Program folder screen (split-display screen)(10.4-inch display unit)

5 Pressing soft key [SINGLE LIST] takes you back to the original program folder display.
Then, the folders are shown which have been selected in the active view for operations on the split display.

Procedure for switching between active folder views for file operations on the split display

On the split folder display, you can switch between active folder views for file operations as described below.
In the active folder view for file operations, the cursor is displayed on the file list.

Procedure

1 When the upper folder view is active for file operations and you press MDI key [SHIFT] and cursor key [↓], the lower folder view becomes active for file operations.

2 When the lower folder view is active for file operations and you press MDI key [SHIFT] and cursor key [↑], the upper folder view becomes active for file operations.

NOTE

1 In step 1, even if you press MDI key [SHIFT] and cursor key [↑] when the upper folder view is active for file operations, the lower folder view becomes active for file operations.

2 In step 2, even if you press MDI key [SHIFT] and cursor key [↓] when the lower folder view is active for file operations, the upper folder view becomes active for file operations.

Explanation

- File operations on the split display

The following operations are allowed on the split display.
<1> You can select and display devices or folders individually in each folder view.
<2> You can copy or move files between folders selected in different folder views.
(For details, see the section on operations of copying/moving between devices.)
In the selected folder view, you can perform the same operations as on the normal screen.

- Path switching

When path switching is performed while the split display is provided for a multi-path system, the folder view in which the CNC_MEM is selected as a device is changed to a view of the last selected folder for each path on the CNC_MEM.

- Device information display

The device information as shown in the normal program folder screen (portion in a dotted box in the Fig. 12.2.4.1 (d)) is not shown on the split display (Fig. 12.2.4.1 (c)).

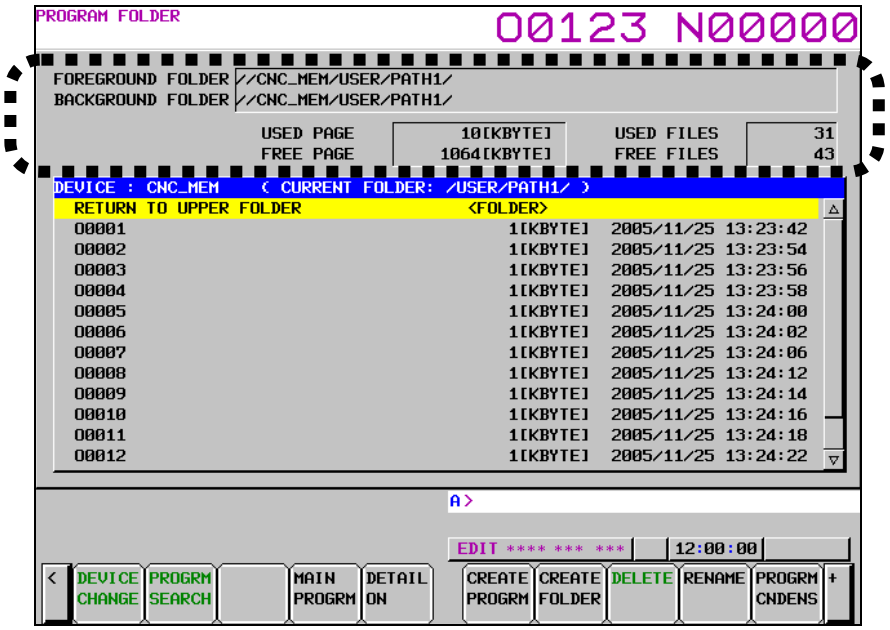


Fig. 12.2.4.1 (d) Program folder screen (normal screen)(10.4-inch display unit)

You should see the device information on the normal screen.

Limitation

- Device that can be selected in both views at the same time on the split display

The same device of any of the following types can be selected in both views at the same time on the split display.

- CNC MEM
- MEM CARD(binary format)
- Data Server

For other types, the same device cannot be selected for display in the folder views at the same time.

If any other type of device is selected when split display starts, the upper and lower folder views cannot show the same sets of folders due to the above limitation. Therefore, the lower folder view shows the folder selected as a foreground folder.

- Display units that support the split display

The split display is supported on 10.4-inch, 15-inch, and 19-inch display units.

12.2.4.2 Folder tree display

Overview

The folder tree can be displayed on program folder screen. By using this function, it is easy to manage programs of each folder.
The folder tree is available in the program memory device (CNC_MEM), the USB device, the memory card program device (MEMCARD), and the data server device.

Explanation

The folder tree is displayed the left side of the screen, and the list of folders and programs is displayed the right side on the screen.
On the folder tree screen, current folder rank and selected folder are displayed.
On the program list screen, programs and folder of selected folder are displayed.

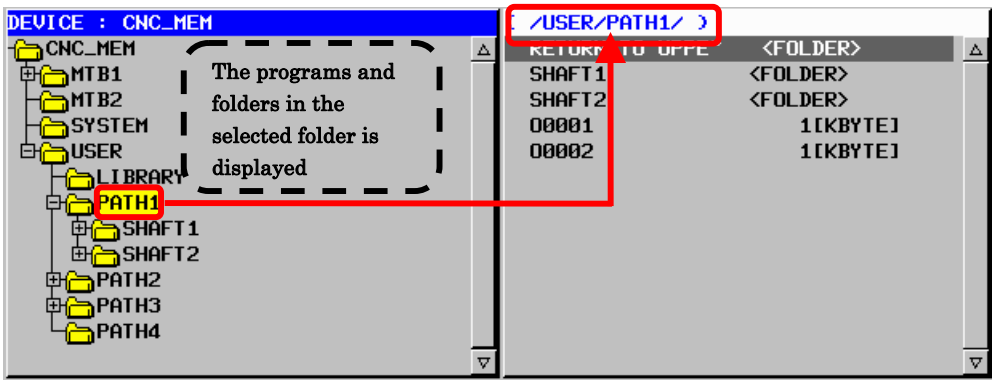


Fig.12.2.4.2 (a) Program folder tree display

Procedure for displaying folder tree

1. Press function key .
2. Press chapter selection soft key [FOLDER].
3. Press the soft key [(OPRT)].
4. Press the soft key [TREE LIST].
5. The folder tree like the above is displayed.
6. If the press the soft key [LIST], the program list screen without folder tree is displayed.

Procedure for changing folder

In the folder tree screen, a folder is changed by moving the cursor.

Selecting upper folder: Press the cursor key .

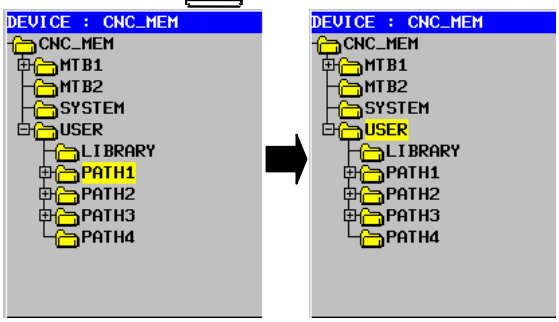


Fig.12.2.4.2 (b)

Select under folder: Press the cursor key .

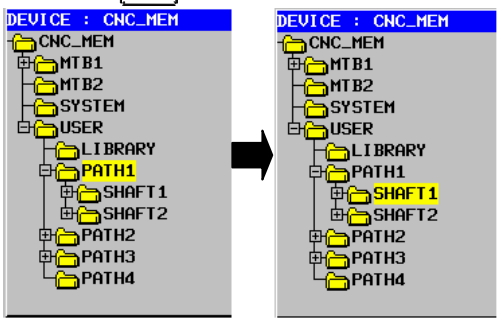


Fig.12.2.4.2 (c)

Select next or back folder: Press the cursor key  or .

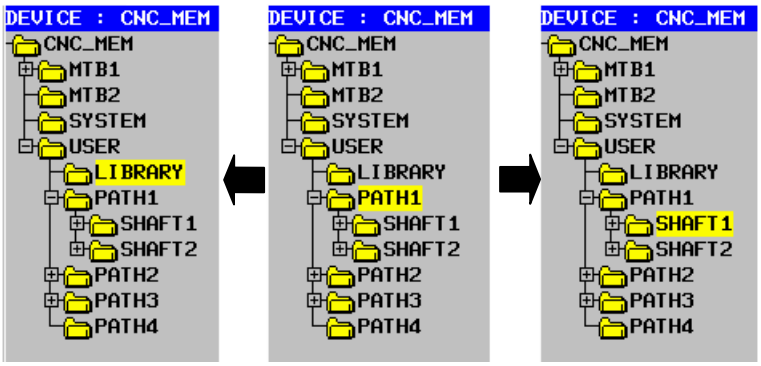


Fig.12.2.4.2 (d)

Limitation

The folder tree is not supported on 7.2/8.4-inch display units.

12.2.5 Next Block Display Screen

Displays the block currently being executed and the block to be executed next.

Procedure for displaying the next block display screen

Procedure

1 Press function key 

.

2 Press chapter selection soft key [NEXT].
The G codes, addresses, command values specified in the block currently being executed and the next block are displayed.

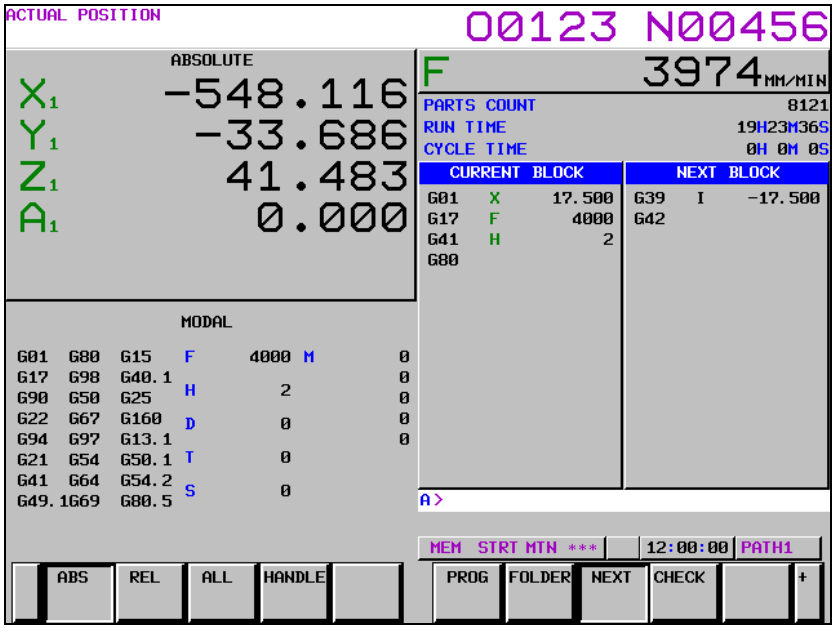


Fig. 12.2.5 (a) Next block display screen (10.4-inch display unit)

12.2.6 Program Check Screen

Displays the program currently being executed, current position of the tool, and modal data.

Procedure for displaying the program check screen

Procedure

1 Press function key 

.

2 Press chapter selection soft key [CHECK].
The program currently being executed, current position of the tool, and modal data are displayed.

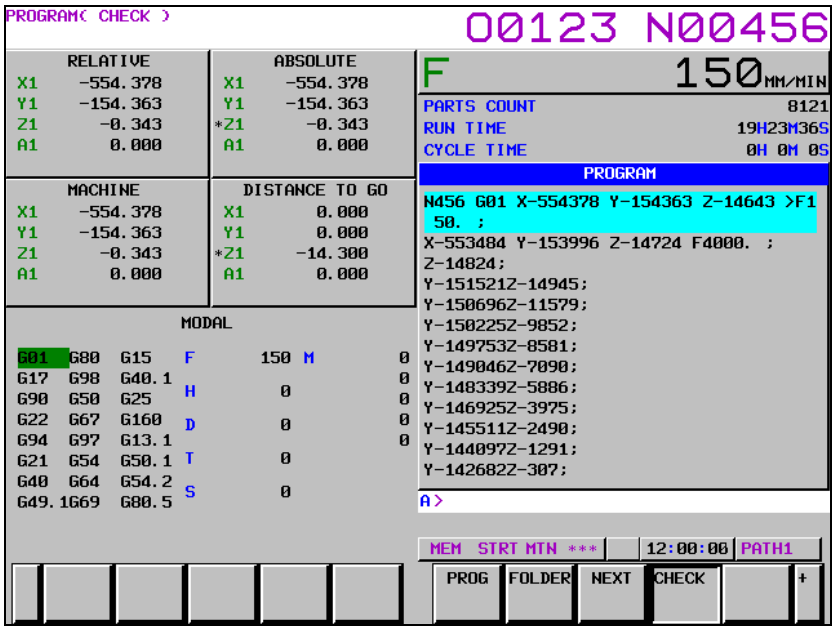


Fig. 12.2.6 (a) Program check screen (10.4-inch display unit)

Explanation

- Program display

The program currently being executed is displayed.
The block being executed is displayed in reverse video.

- Current position display

The current position in the relative coordinate system, workpiece coordinate system, and machine coordinate system, and the remaining amount of travel are displayed.

- Modal G codes

Up to 24 modal G codes are displayed.

12.2.7 Background Editing

Editing one program during execution of another program is referred to as background editing. You can perform the same edit operations in the background as those in normal editing (foreground editing).

On a 10.4-inch or 15-inch display, you can perform background editing for the programs displayed in multiple tiled windows. You can switch from a program displayed in one of the tiled windows to another to perform copy, paste, and other edit operations, which allows you to edit programs efficiently.
You can edit one program simultaneously on a 8.4-inch display or three programs on a 10.4-inch display.

Function

- Background editing

A program other than the currently selected program can be edited. Background editing can be performed in any mode.

- EDIT mode and reference mode

When background editing is started in the EDIT mode, the program can be edited. When background editing is started in the reference mode, the program is opened in the read-only mode and cannot be edited.
While a program opened in the EDIT mode cannot be run, a program opened in the reference mode can be run.

- Program selected in the foreground

If the program selected in the foreground is specified as a program to be edited in the background, background editing is started in the read-only mode. The text at an arbitrary position of the program currently being executed can be checked.

- Switching from the program directory screen to the editing screen

By selecting a program from the program folder screen, background editing can be started.
On the program folder screen, move the cursor to a program to be edited in the background, then press the MDI key 

. The screen display switches to the background editing screen.
Background editing can be started without entering a program name.

- Background editing on a multi-path system

Background editing does not depend on the path.
Even if the path to be operated is changed using the path selection signal, the background editing currently being executed is just continued.

NOTE
The following programs cannot be edited in the background. They can be opened only in the read-only mode.
- Running program
- Main program
- Program with the edit disable attribute

Display

When background editing starts, the ordinary program editing screen switches to the background editing screen.
When two or more programs are edited in the background, the screen is split to display these programs. For a 10.4-inch display, you can edit two programs simultaneously; for a 10.4-inch display, you can edit three programs simultaneously.

- Word editing

Fig. 12.2.7 (a) shows background word editing performed simultaneously for two programs (right and left programs). On the status line at the top of the window for each program, the program name and "BG-EDIT" (indicating background editing in progress) are displayed.
The status line of the program being edited is displayed in reverse video.

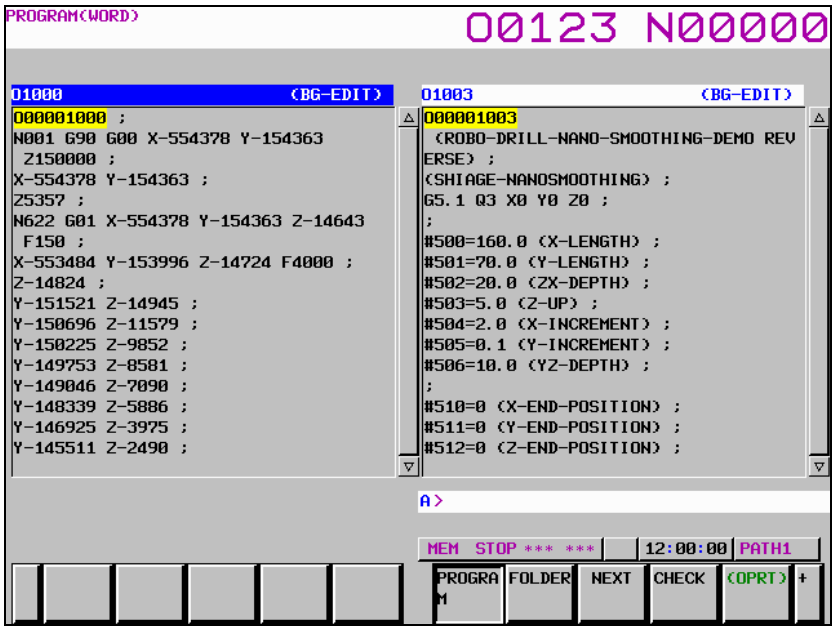


Fig. 12.2.7 (a) Background editing screen (word editing) (10.4-inch display unit)

- Character editing

Fig. 12.2.7 (b) shows background character editing performed simultaneously for two programs (right and left programs). Similarly to word editing, at the top of the window for each program, the status line is displayed. In addition, the current input mode (INSERT MODE or OVERWRITE MODE) is displayed at the upper right of the screen for character editing.
The status line of the program being edited is displayed in reverse video.

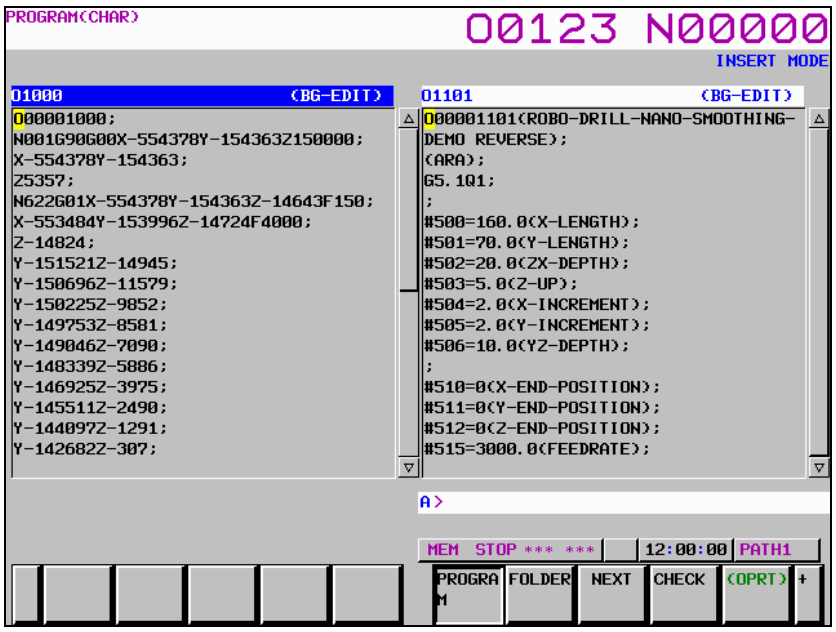


Fig. 12.2.7 (b) Background editing screen (character editing) (10.4-inch display unit)

- Editing status

The following items are displayed on the status line and program editing area according to the background editing status.

Editing status	Displayed items
No program selected	(BG-EDIT) “NO PROGRAM” is displayed in the editing area.

Editing status	Displayed items
Program opened	program-name + (BG-EDIT)
Read-only program opened	program-name + (BG:READ ONLY) The contents of the program are displayed in green.

Starting background editing from the editing screen

Procedure (word editing)

Method 1

1 Press function key .
2 Press soft key [PROGRAM].
3 Enter a program name.
4 Press soft key [BG EDIT].

NOTE
1 Background editing is performed in the EDIT mode.
2 If a nonexistent program is specified, a new program is created.

Method 2

1 Press function key .
2 Press soft key [PROGRAM].
3 Press soft key [(OPRT)], then soft key [BG EDIT].
4 Enter a program name.
5 Press soft key [EDIT EXEC] to start background editing in the EDIT mode or soft key [REF EXEC] to start background editing in the reference mode.

NOTE
If a nonexistent program is specified,
- Pressing soft key [EDIT EXEC] causes a new program to be created.
- Pressing soft key [REF EXEC] causes the warning message “SPECIFIED PROGRAM NOT FOUND” to appear. A new program is not created.

Method 3

1 Press function key .
2 Press soft key [PROGRAM].
3 Press soft key [(OPRT)], then soft key [BG EDIT].
4 Press soft key [EDIT EXEC] to start background editing in the EDIT mode or soft key [REF EXEC] to start background editing in the reference mode. At this time, “NO PROGRAM” is displayed on the screen.
5 Enter a program name.
6 Press soft key [PROGRM SEARCH].

NOTE
If a nonexistent program is specified, the warning message “SPECIFIED PROGRAM NOT FOUND” is displayed. A new program is not created.

Procedure (character editing)

Method 1

1 Press function key .

2 Press soft key [PROGRAM].
3 Press soft key [(OPRT)], then soft key [BG EDIT].
4 Enter a program name.
5 Press soft key [EDIT EXEC] to start background editing in the EDIT mode or soft key [REF EXEC] to start background editing in the reference mode.

NOTE
If a nonexistent program is specified,
- Pressing soft key [EDIT EXEC] causes a new program to be created.
- Pressing soft key [REF EXEC] causes the warning message “SPECIFIED PROGRAM NOT FOUND” to appear. A new program is not created.

Method 2

1 Press function key 

.
2 Press soft key [PROGRAM].
3 Press soft key [(OPRT)], then soft key [BG EDIT].
4 Press soft key [EDIT EXEC] to start background editing in the EDIT mode or soft key [REF EXEC] to start background editing in the reference mode. At this time, “NO PROGRAM” is displayed on the screen.
5 Press soft key [PROGRM SEARCH].
6 Enter a program name.
7 Press soft key [PROGRM NAME].

NOTE
If a nonexistent program is specified, the warning message “SPECIFIED PROGRAM NOT FOUND” is displayed. A new program is not created.

When background editing has already started and editing start operation is performed again, the program being edited and newly specified program are edited simultaneously.

Starting background editing from the program folder screen

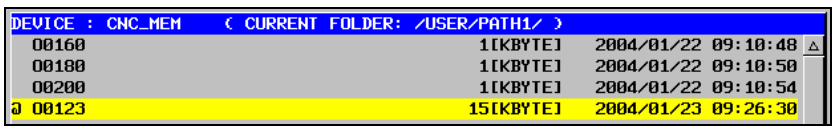
By selecting a program from the program folder screen, background editing can be started. The cursor is used to select a program. You do not need to enter a program name.

Procedure

1 Press function key 

.
2 Press soft key [FOLDER].
3 Move the cursor to a program to be edited.
4 Press the MDI key 

. The screen display is switched to the background editing screen.



NOTE
When background editing is started from the program folder screen, the EDIT mode is set.
For the following programs, the reference mode is set, however:
- Running program
- Main program
- Program with the edit disable attribute

Background editing operation

- Editing operation

The same editing operations as performed in the foreground can be performed.

- Switching from a program to another for editing

To switch from a program to another for editing when editing multiple programs simultaneously in the background, press 

key, then  or .

- Editing operation in the read-only mode

Cursor movement and page switching are possible. However, program editing is disabled.

- Copying and pasting text between programs

Text in a program selected and copied during program editing is kept held after the program to be edited is changed. You can paste text in a program onto another program.

Background editing end operation

Background editing can be ended using the procedure described below.
The procedure for ending background editing of one program and that for ending all background editing of multiple programs are shown below.

- Ending editing of one program

1 Press function key 

.
2 Press soft key [PROGRAM].
3 Select a program for which to end editing.
4 Press soft key [(OPRT)], then soft key [BG END].

- Ending editing of all programs

1 Press function key 

.
2 Press soft key [PROGRAM].
3 Press soft key [(OPRT)], then soft key [BG ALL END].

To return to ordinary foreground editing, end all background editing. If at least one program is being edited in the background, you cannot return to foreground editing.

12.2.8 Stamping the Machining Time

The execution times of the most recently executed ten programs can be displayed in hours, minutes, and seconds.
The calculated machining time can be inserted as a comment of the program to check the machining time on the program folder screen.

Procedure for stamping the machining time

Procedure

- Displaying the machining time

1 Press function key [PROG].

2 Press the continuous menu key [▷] several times to display soft key [TIME].

3 Press soft key [TIME]. The machining time display screen appears.

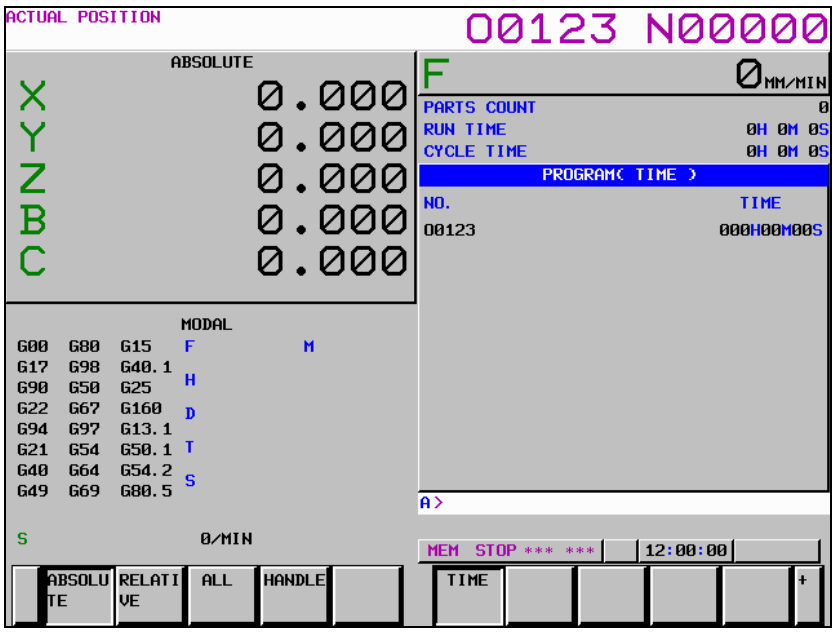


Fig. 12.2.8 (a) Machining time display screen (10.4-inch display unit)

- Calculating the machining time

1 Select the memory operation mode, then press the [RESET] key.

2 Select the program screen, then select a program of which machining time you want to calculate.

3 Execute the program to perform actual machining.

4 When the [RESET] key is pressed or, M02 or M30 is executed, the machining time count operation stops. When the machining time display screen is selected, the program number of the stopped main program and its machining time are displayed.

5 The Fig. 12.2.8 (b) shows the screen when the machining times of the ten main programs O0020, O0040, …, and O0200 are displayed and the screen when the machining time of O0220 is newly calculated after that.

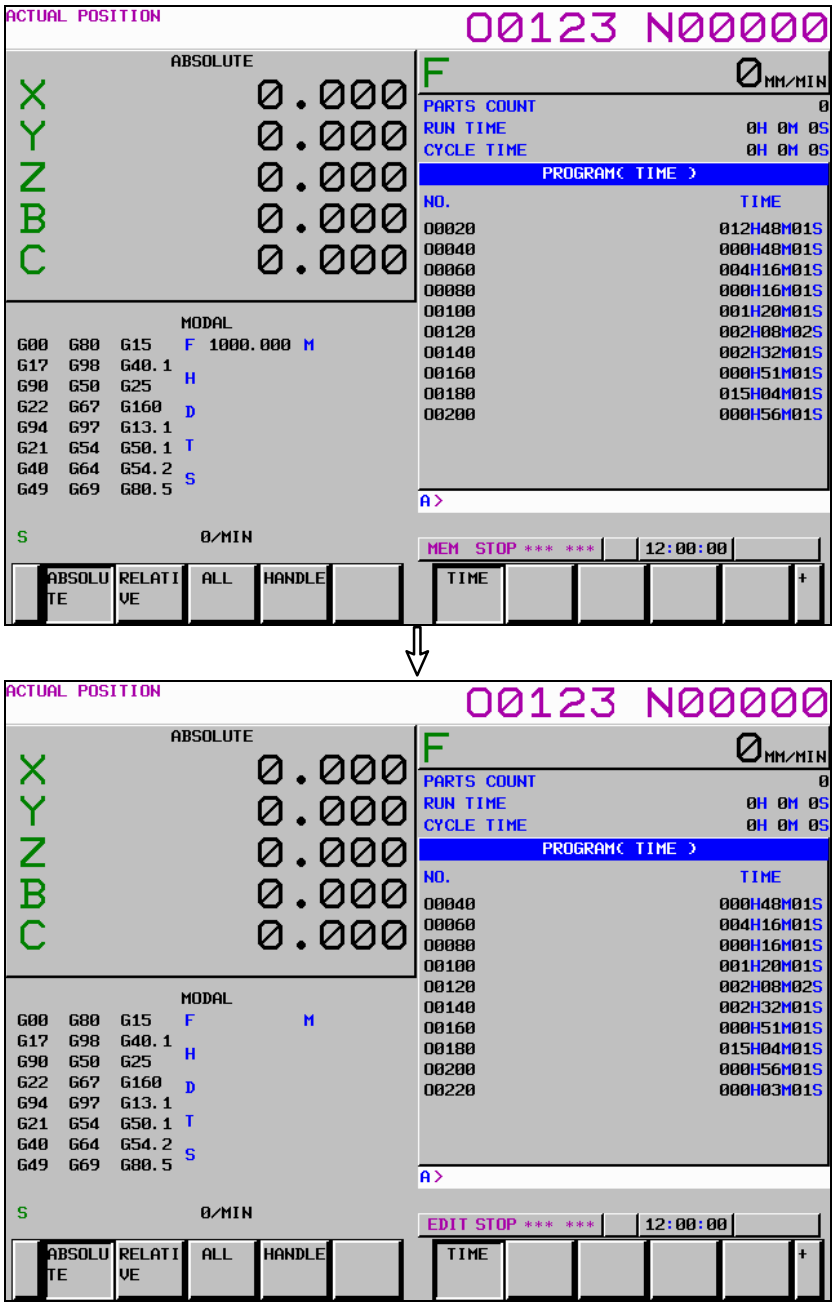


Fig. 12.2.8 (b) Stamping the machining time (10.4-inch display unit)

Procedure for inserting the machining time on the program screen

Procedure

You can display the machining time of a program as a comment of the program. The procedure is shown below:

1 To insert the calculated machining time of a program as a comment, the machining time of the program must be displayed on the machining time display screen. Before stamping the machining time of the program, check that the program number is displayed on the machining time display screen.
2 Set the edit mode or background editing state and select the program screen. Then, select a program of which machining time you want to insert.
3 Suppose that the machining time of O0100 is displayed on the machining time display screen.

Press soft key [(OPRT)] to display the operation soft keys. Then, press the continuous menu key ▷ several times to display soft key [INSERT TIME]. Press soft key [INSERT TIME]. The beginning of the program is displayed and the machining time of the program is inserted after the program name.

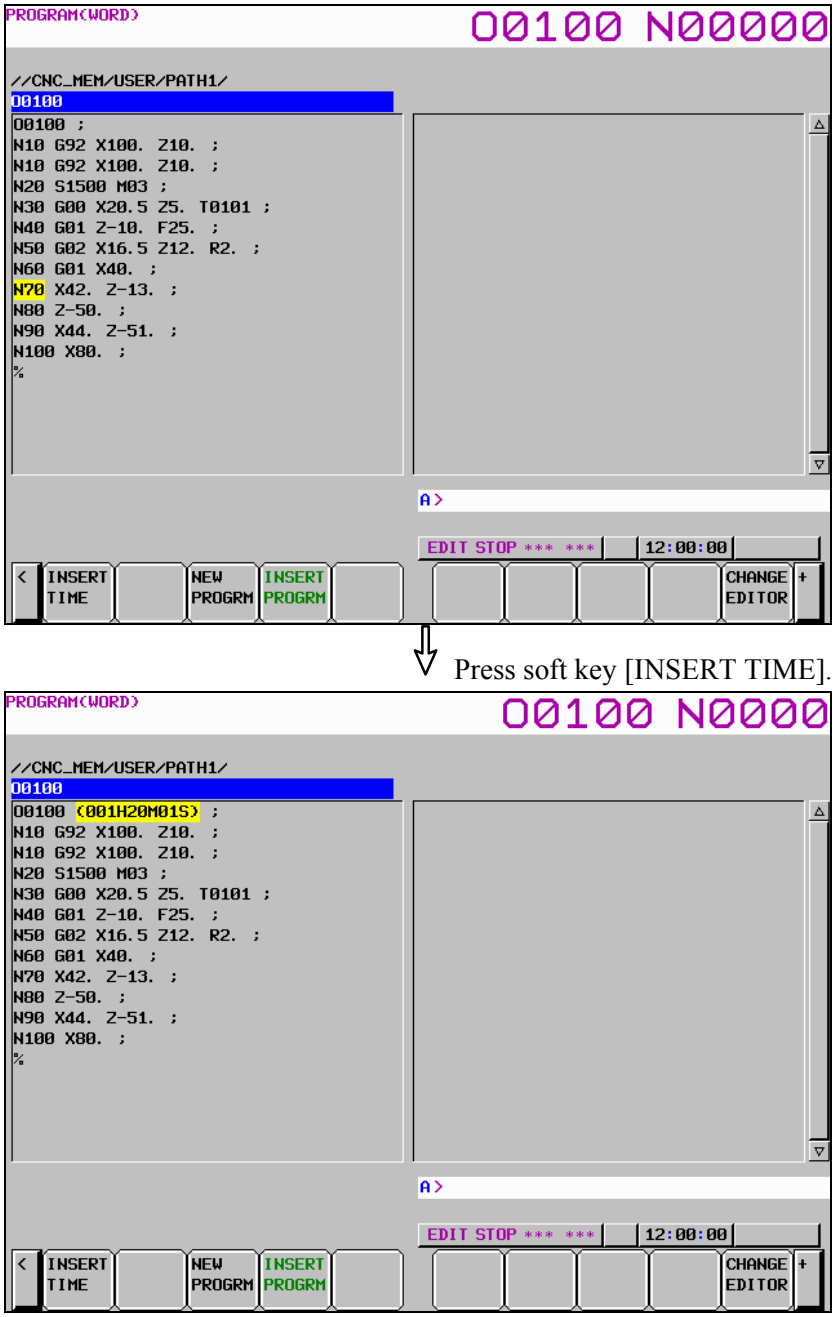


Fig. 12.2.8 (c) Program screen (10.4-inch display unit)

4 If a comment is written in the block containing the program number of a program of which machining time is to be inserted, the machining time is inserted after the comment.

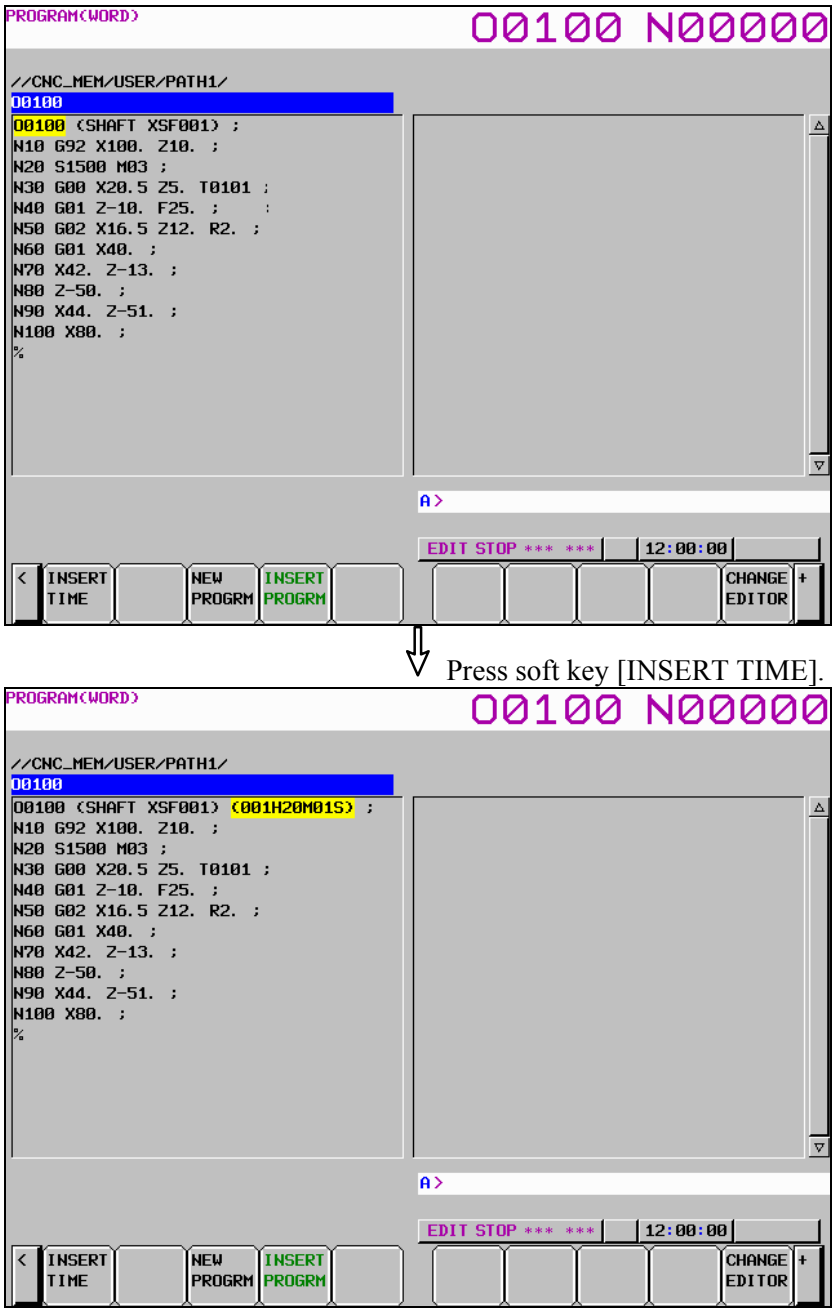


Fig. 12.2.8 (d) Program screen (10.4-inch display unit)

Display on the program folder screen

The machining time of a program inserted in the program as a comment is displayed after the existing comment of the program on the program folder screen.

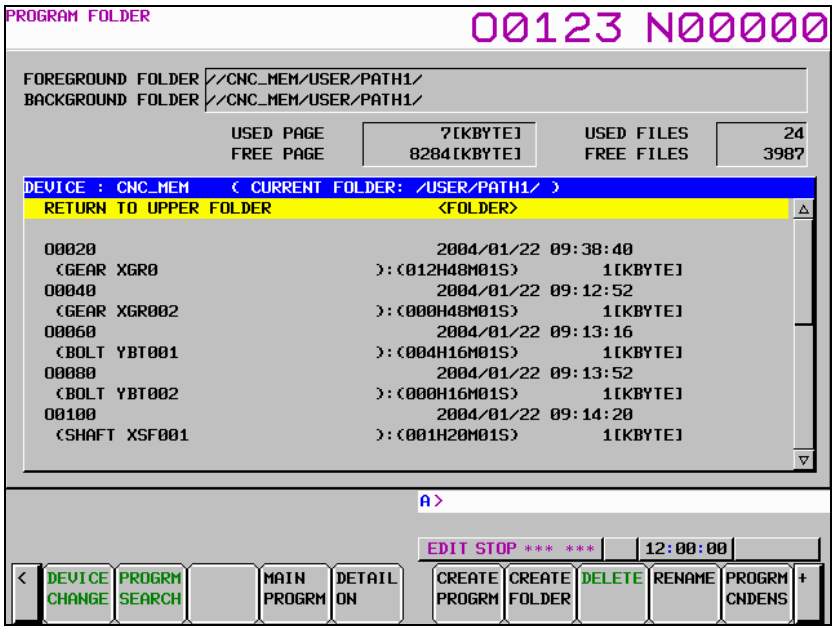


Fig. 12.2.8 (e) Program folder screen (10.4-inch display unit)

Explanation

- Machining time

The machining time is counted from the initial start after a reset in the memory operation mode to the next reset. If a reset is not performed during operation, the machining time is counted from the start to M02 or M30. The time during which operation is held is not counted, but the time used to wait for the completion of the M, S, T, and/or B functions is counted, however.

- Stamping the machining time

You can insert (stamp) the displayed machining time in a program stored in memory as a comment. The machining time is inserted as a comment after the program number.

- Program folder

You can display the machining time inserted after a program number on the program folder screen. This operation lets you know the machining time of each program to use the time as useful reference data when planning processes on site.

Limitation

- Alarm

The execution of a program may be held by an alarm during the machining time count operation. In this case, the machining time is counted until the alarm is released by a reset.

- M02

It may be specified that M02 does not reset the CNC, but returns completion signal FIN to the CNC to restart the program from the beginning successively (bit 5 (M02) of parameter No. 3404 is set to 0). In this case, when M02 returns completion signal FIN, the machining time count operation stops.

- Stamping the machining time

When an attempt is made to stamp the machining time of a program in the program, the machining time of the program may not be displayed on the machining time display screen. In this case, the machining time cannot be inserted in the program even if soft key [INSERT TIME] is pressed.

- Correcting the machining time

If an incorrect machining time is calculated (such as when a reset is made during the execution of a program), reexecute the program to calculate the correct machining time. The same program number may be displayed in two or more lines on the machining time display screen. In this case, the latest calculated machining time is inserted in the program.

- States of the stamped machining time

In the following states, the stamped machining time is displayed on the program folder screen as shown below.

1 When the comment of a program is longer than 16 characters
The 17th and subsequent characters of the comment are discarded and the machining time display field is left blank.

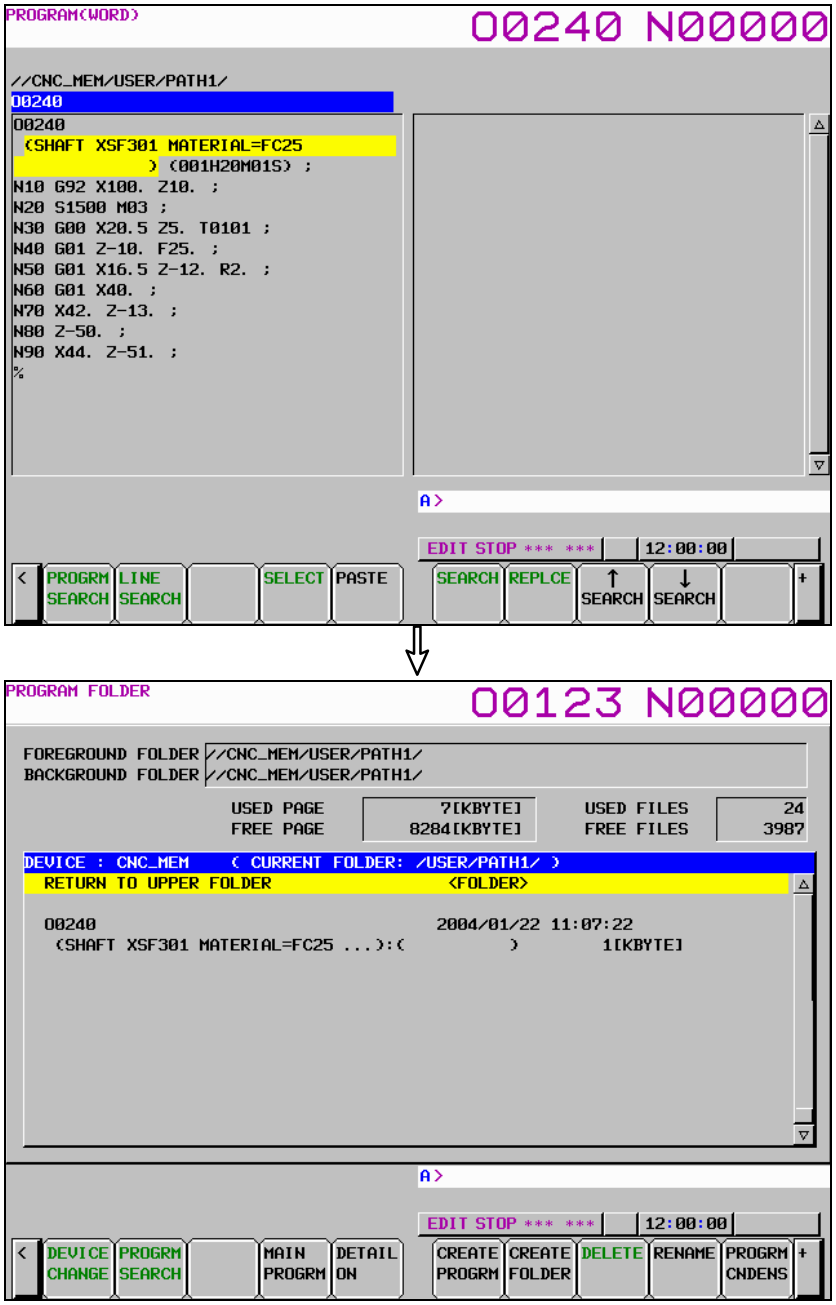


Fig. 12.2.8 (f) 1. When the comment of a program is longer than 16 characters (10.4-inch display unit)

2 When two or more machining times are stamped
The first machining time is displayed.

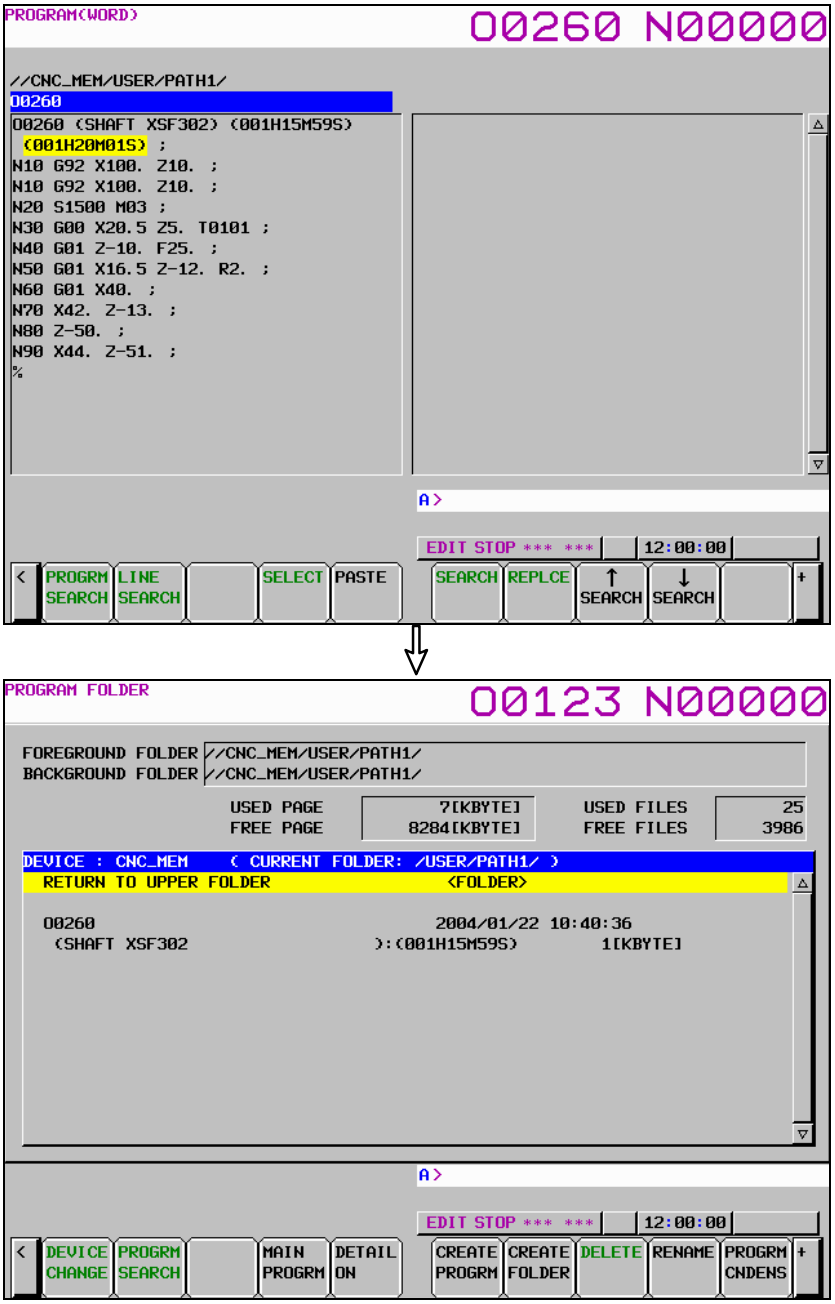


Fig. 12.2.8 (g) When two or more machining times are stamped (10.4-inch display unit)

3 When the format of an inserted machining time is not “hhhHmmMssS” (H following a 3-digit number, M following a 2-digit number, and S following a 2-digit number, in this order)
The machining time display field is left blank.

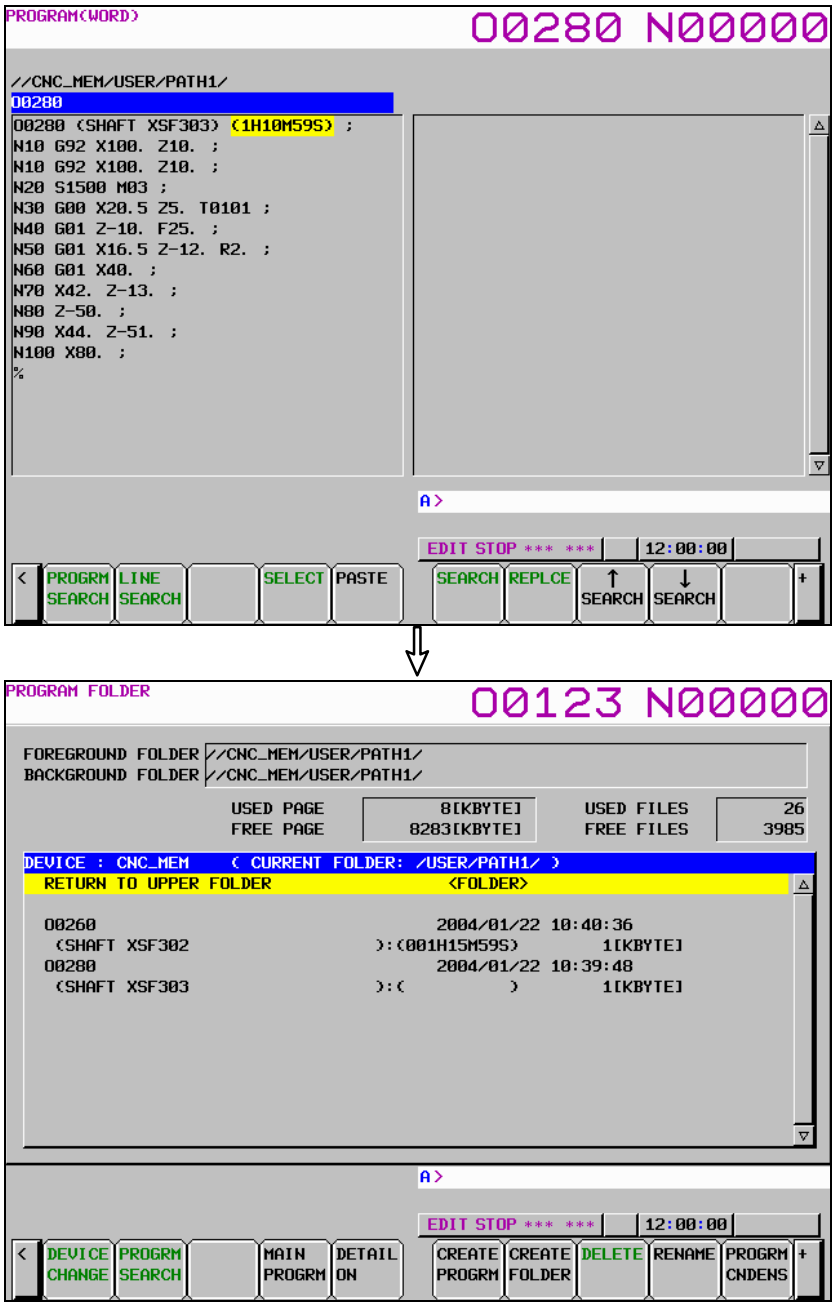


Fig. 12.2.8 (h) When the format of an inserted machining time is not "hhhHmmMssS" (H following a 3-digit number, M following a 2-digit number, and S following a 2-digit number, in this order) (10.4-inch display unit)

12.2.9 Screen for Assistance in Entering Tilted Working Plane Indexing

The screens for assistance in entering tilted working plane indexing (hereinafter referred to as guidance screens) include a command type selection screen and a tilted working plane data setting screen. The command type selection screen is used to select a tilted working plane indexing. The tilted working plane data setting screen is used to set specified tilted working plane data required for the selected command. By making settings and performing operations on these guidance screens, a tilted working plane indexing block can be created.

The created block is reflected as a new insertion to a program being edited or as a modification to an existing block.

This function can be enabled by setting bit 1 (GGD) of parameter No. 11304 to 1.

Creation of a new block

The following describes the procedure for creating a tilted working plane indexing block on guidance screens and for inserting the block to a program being edited on a program editing screen.

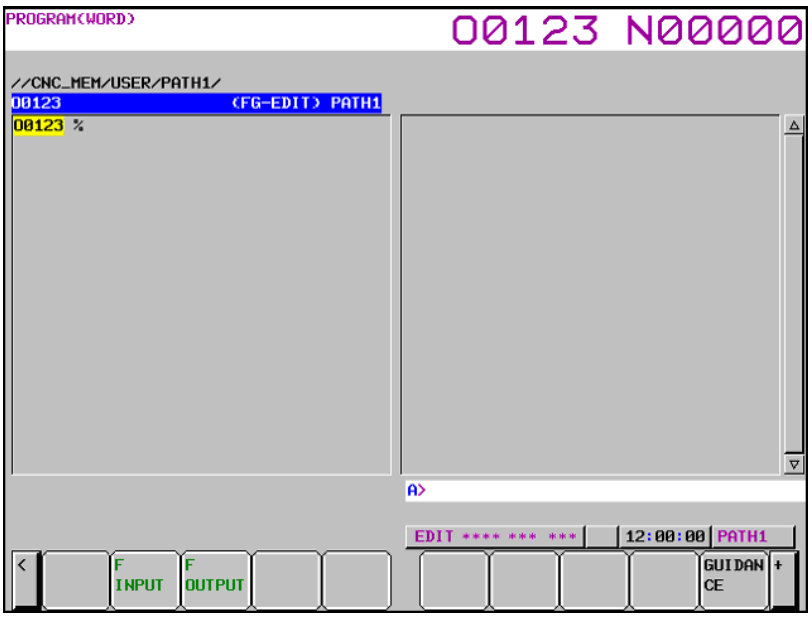
1 On a program editing screen, display a program to which you want to insert a tilted working plane indexing block.
The foreground editing screen, background editing screen, or MDI editing screen should be displayed.

- Displaying the foreground editing screen
  <1> Select the EDIT mode.
  <2> Press function key PROG.
  <3> Press soft key [PROGRAM].

- Displaying the background editing screen
  <1> Press function key PROG.
  <2> Press soft key [FOLDER].
  <3> Press any of the cursor keys to move the cursor to a program to be edited in the background.
  <4> Press INPUT key.

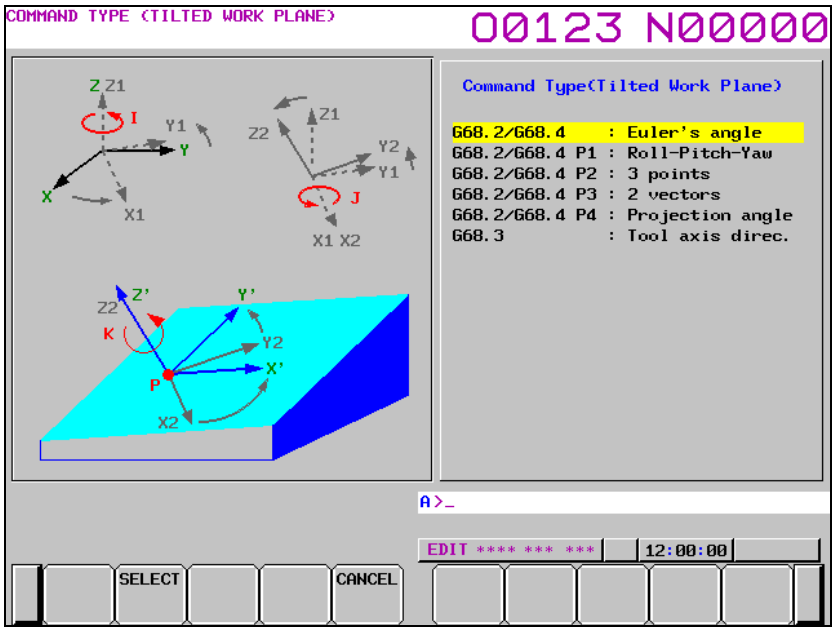
- Displaying the MDI editing screen
  <1> Select the MDI mode.
  <2> Press function key PROG.
  <3> Press soft key [PROGRAM].

The program editing screen is displayed.

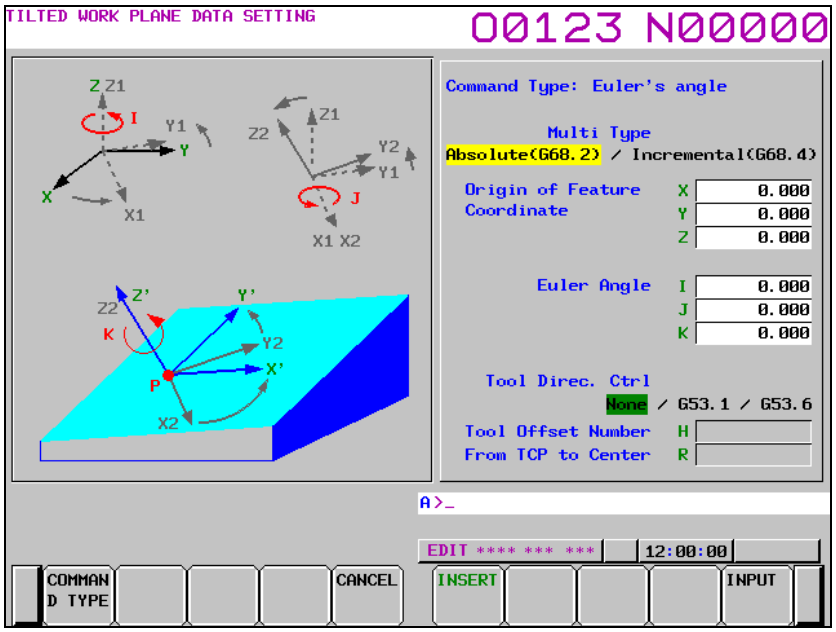


2 Press any of the cursor keys to move the cursor to a position where you want to insert a block.
Note that a block created on the guidance screens is inserted after the block at the cursor position. (If the block at the cursor position includes a tilted working plane indexing, the existing block is modified. See "Modification to an existing block" below.)

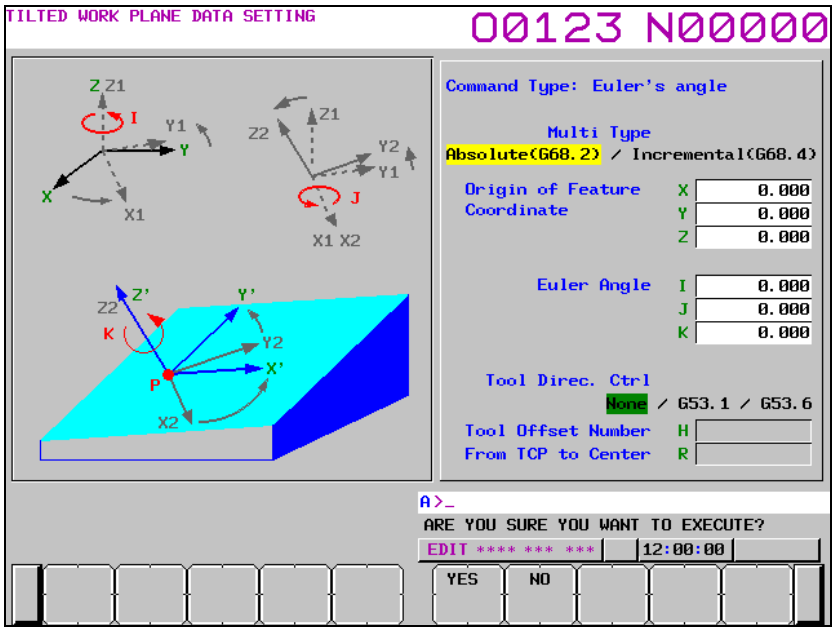
3 Press soft key [(OPRT)].
4 Press continuous menu key ▷ several times, and then press soft key [GUIDANCE].
The command type selection screen is displayed.



5 Select a command type with any of the cursor keys, and then press soft key [SELECT].
The tilted working plane data setting screen is displayed.



6 Enter command data for the setting items.
7 Press soft key [INSERT].

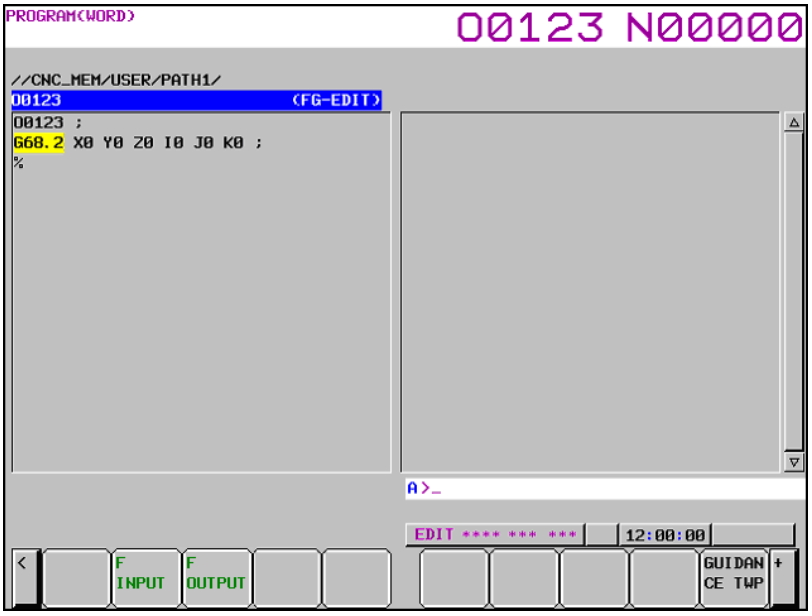


8 Press soft key [YES].
This takes you back to the program editing screen, where the new block is inserted after the block at the cursor position.

Modification to an existing block

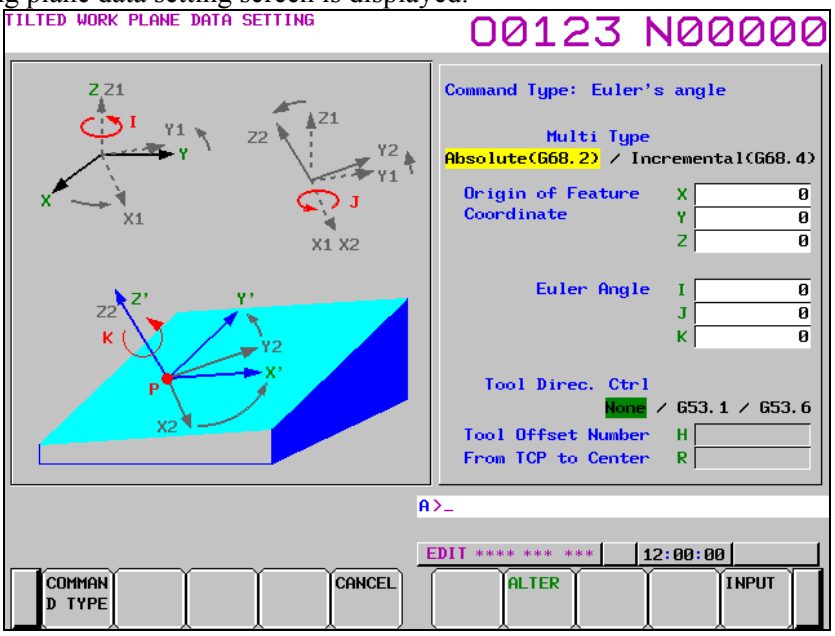
The following describes the procedure for replacing a block in a program being edited on a program editing screen, with a tilted working plane indexing block created on a guidance screen.

1 On a program editing screen, display a program to be edited.
(For the procedure for displaying a program editing screen, see step 1 in "Creation of a new block".)
The program editing screen is displayed.



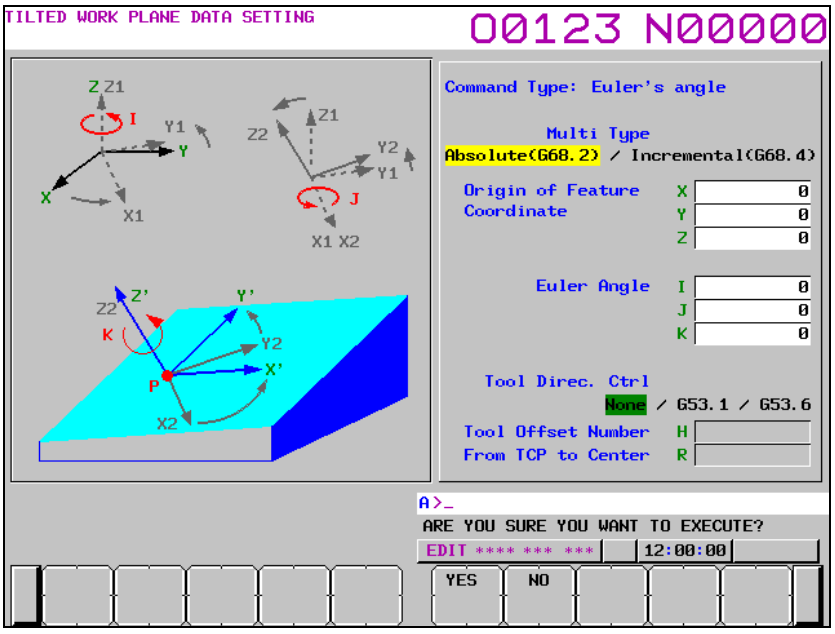
2 Press any of the cursor keys to move the cursor to a block to be modified.
For a command extending over more than one block, move the cursor to the first block.
3 Press soft key [(OPRT)].

4 Press continuous menu key ▷ several times, and then press soft key [GUIDANCE TWP]. The tilted working plane data setting screen is displayed.



5 Enter command data for setting items to be modified.

6 Press soft key [ALTER].



7 Press soft key [YES].
This takes you back to the program editing screen, where the block at the cursor position is replaced.

Guidance screen cancellation

Pressing soft key [CANCEL] on a guidance screen takes you back to the program editing screen. At this time, the data that has been set on the guidance screen is discarded.

NOTE

1 In addition to the above operation, the following operations also cancel a guidance screen. The data that has been set on the guidance screen is discarded.
- When bit 7 (CPG) of parameter No. 11302 is 1 (setting for automatically switching between program-related screens according to CNC mode), the CNC mode is changed.
- When a guidance screen is displayed from the foreground editing screen, the CNC mode is changed to a mode other than EDIT, TJOG, or THND.
- When a guidance screen is displayed from the MDI editing screen, the CNC mode is changed to a mode other than MDI.
- When a guidance screen is displayed on a 15/19-inch display unit, the screen is switched by a vertical soft key.
- The screen is switched by an MDI key.
- The screen is switched by a path select signal.
- An event that causes screen switching has occurred, including occurrence of an alarm, display of an operator message, or signal-based display of a screen (such as a tool compensation screen, workpiece shift screen, workpiece coordinate system setting screen, or C Language Executor screen).

2 If MDI key 

is pressed after switching from a guidance screen to another screen, a program editing screen is displayed, instead of the guidance screen.

Notes

- Conditions under which soft key [GUIDANCE TWP] is displayed
  Soft key [GUIDANCE TWP] is displayed on a program editing screen under the following conditions:
  1 Foreground editing screen
    - The CNC mode is EDIT, TJOG, or THND.
    - The editing and display are not prohibited for a program to be edited.
  2 Background editing screen
    - A program to be edited is not in reference mode.
    - The editing and display are not prohibited for a program to be edited.
  3 MDI editing screen
    - The CNC mode is MDI.

- Screen displayed when soft key [GUIDANCE TWP] is pressed
  When soft key [GUIDANCE TWP] is pressed, which guidance screen is displayed is automatically determined according to the conditions described below.
  1 When a tilted working plane indexing is not included in a block at the cursor position on a program editing screen
    The command type selection screen is displayed. A block created on the guidance screens is inserted after the block at the cursor position on the program editing screen.
  2 When a tilted working plane indexing is included in a block at the cursor position on a program editing screen

The tilted working plane data setting screen is displayed, which shows the data for the tilted working plane indexing in the block at the cursor position on the program editing screen.
When a block of a tool axis direction control command exists immediately after the tilted working plane indexing, the command data for the block is also shown. The tilted working plane indexing in the block at the cursor position on the program editing screen is replaced with a block created on the guidance screen.

NOTE
If the CNC is in the reset state or emergency stop state when soft key [GUIDANCE TWP] is pressed on the foreground editing screen or MDI editing screen, the warning "PROGRAM READ FAILED" appears, and the operation cannot be continued. (Soft keys other than [CANCEL] are not displayed.) Press soft key [CANCEL] to return to the program editing screen, and then press soft key [GUIDANCE TWP] again.

12.2.9.1 Command type selection screen

The command type selection screen is used to select the type of a tilted working plane indexing you want to insert into a program to be edited. One of the following command types can be selected:

- G68.2 / G68.4 (Euler's angle)
- G68.2 / G68.4 P1 (Roll-Pitch-Yaw angle)
- G68.2 / G68.4 P2 (3 points specification)
- G68.2 / G68.4 P3 (2 vectors specification)
- G68.2 / G68.4 P4 (Projection angle)
- G68.3 (Tool Axis Direction)

G68.2 is an absolute command, and G68.4 is an incremental command.

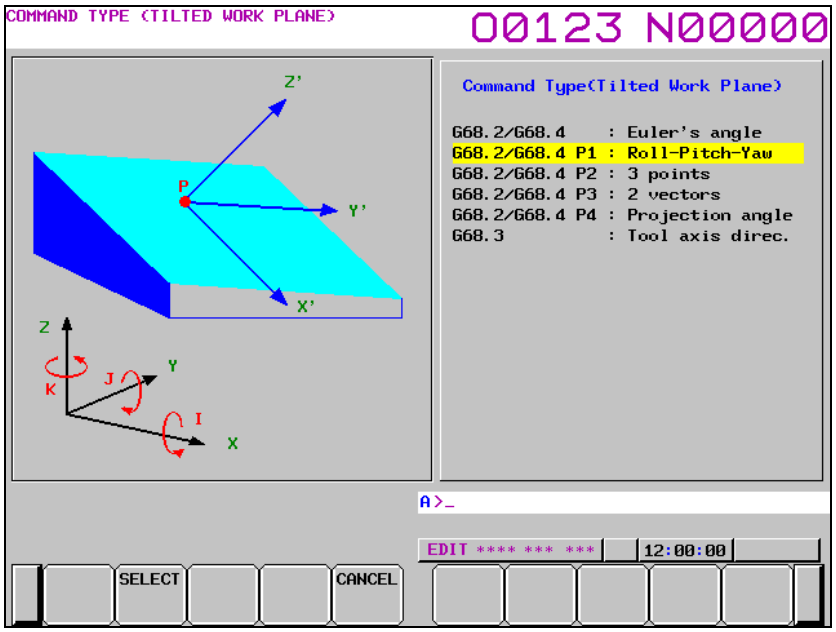


Fig. 12.2.9.1 (a) Command type selection screen (10.4-inch display unit)

Selection of a command type

1 Press cursor key ↑ or ↓ to move the cursor to a command type you want to select. As the cursor is moved, the figure corresponding to the command type at the cursor position is displayed.

2 When soft key [SELECT] is pressed, the command type at the cursor position is accepted, and the tilted working plane data setting screen is displayed.

NOTE
If the warning "PROGRAM READ FAILED" appears when the command type selection screen is displayed, the operation cannot be continued. (Soft keys other than [CANCEL] are not displayed.) Press soft key [CANCEL] to return to the program editing screen, and then press soft key [GUIDANCE TWP] again.

12.2.9.2 Tilted working plane data setting screen

The tilted working plane data setting screen is used to set specified tilted working plane data required for a tilted working plane indexing of the type that has been selected on the command type selection screen or has been selected when soft key [GUIDANCE TWP] is pressed.
Different types of tilted working plane data setting screen are provided for different command types. For details of each type of tilted working plane data setting screen, see "Details of the tilted working plane data setting screen".

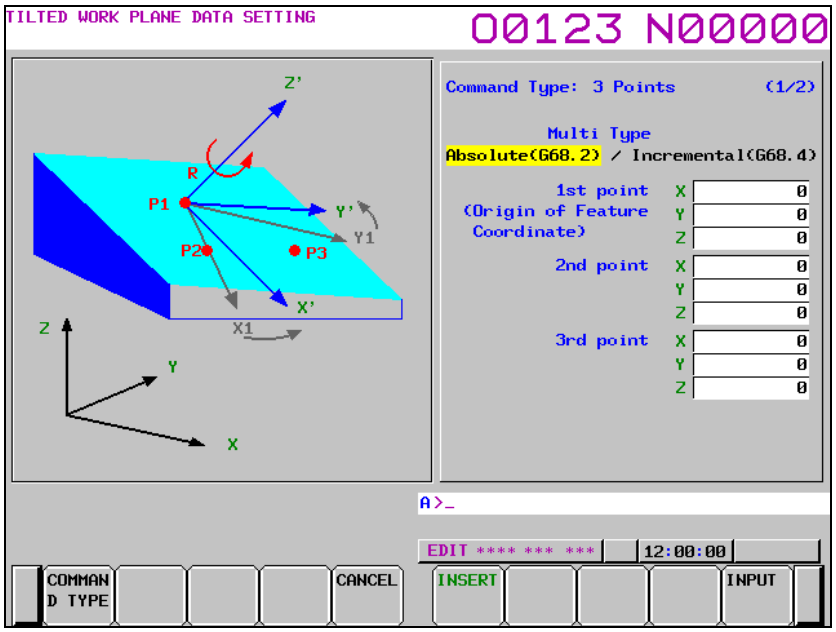


Fig. 12.2.9.2 (a) Tilted working plane data setting screen-3 points specification(10.4-inch display unit)

Display of the tilted working plane data setting screen

When a new block is created, the initial values are set in the setting and selection items.
When an existing block is modified, the command data for the block at the cursor position on the program editing screen is reflected in the setting and selection items.

NOTE
1 For existing block modification, if the guidance screen is displayed when the cursor is placed in the middle of multiple blocks for a command, the parameters for the block(s) before the cursor are not reflected in the setting and selection items, but only the parameters for the block(s) after the cursor are reflected. If no data is to be reflected in a setting or selection item, the initial value is set in the item.
2 For existing block modification, only the commands with parameter-specified axis names are reflected in the setting items on the screen. (The values specified in data for commands with incorrect axis names are not reflected in the setting items.)

Command data input

- Item for which to enter a value

  Press cursor key  or  to move the cursor to an item you want to set.

  Enter a value, and then press the  key or soft key [INPUT].



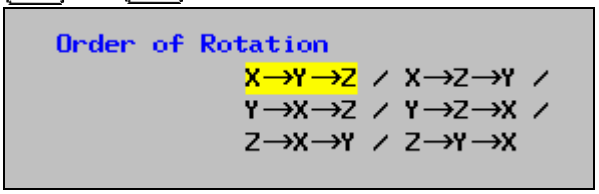
  Example)
  When the origin of a feature coordinate system is set as shown above, addresses X, Y, and Z are specified as follows:
  G68.2 X0.001 Y0.01 Z1000 ...

  For two items "Tool Offset Number" and "From TCP to center", the state with no command can be set by deleting the specified values according to the following procedure.

  1 Press cursor key  or  to move the cursor to the item "Tool Offset Number" or "From TCP to Center".
  2 Press  or soft key [INPUT] without entering anything.

- Item to be selected from a list

  1 Press cursor key  or  to move the cursor to an item you want to set.
  2 Press cursor key  or  to move the cursor to an item you want to select.



  Example)
  Order of rotation for Roll-Pitch Yaw angle

Insertion of a block

If the guidance screen is displayed when the block at the cursor position on the program editing screen does not include a tilted working plane indexing, soft key [INSERT] is displayed on the tilted working plane data setting screen. An operation for inserting a tilted working plane indexing block is described below.
For warnings that may be issued at the time of block insertion, see "Limitation".

1 Press soft key [INSERT].
The confirmation message "ARE YOU SURE YOU WANT TO EXECUTE?" is displayed, and soft keys [YES] and [NO] are displayed.
2 Press soft key [YES].
A block is created from the command type and command data and then is inserted after the block at the cursor position in the program to be edited. After block insertion, the guidance screen is closed and the program editing screen is displayed.
Pressing soft key [NO] takes you back to the tilted working plane data setting screen.

Example)
G00 X0.; When the guidance screen is displayed and the 3-point specification is selected as a
↓ command type for block insertion, a created block is inserted after the block at the cursor position.
G00 X0.;
G68.2 P2 Q0...
G68.2 P2 Q1...
G68.2 P2 Q2...
G68.2 P2 Q3...

Replacement of a block

If the guidance screen is displayed when the block at the cursor position on the program editing screen includes a tilted working plane indexing, soft key [ALTER] is displayed on the tilted working plane data setting screen. An operation for replacing a tilted working plane indexing block is described below.
For warnings that may be issued at the time of block replacement, see "Limitation".

1 Press soft key [ALTER].
The confirmation message "ARE YOU SURE YOU WANT TO EXECUTE?" is displayed, and soft keys [YES] and [NO] are displayed.
2 Press soft key [YES].
A block is created from the command type and command data and then replaces the tilted working plane indexing in the block at the cursor position on the program editing screen. After block replacement, the guidance screen is closed and the program editing screen is displayed.
Pressing soft key [NO] takes you back to the tilted working plane data setting screen.

NOTE
1 If a block to be replaced includes a command other than a tilted working plane indexing, the command is deleted during block replacement. However, only the sequence number at the beginning is preserved.
2 If the guidance screen is displayed when the cursor is placed in the middle of multiple blocks for a command, the block(s) before the cursor are not replaced. These blocks remain unchanged after replacement.
3 If soft key [GUIDANCE TWP] is pressed when the cursor is placed on a block of a tool axis direction control command, the command type selection screen is displayed in new insertion mode. A block created on the guidance screens is inserted after the block of the tool axis direction control command.

Limitation

The following lists the warnings that may be issued at the time of block insertion or replacement.
If a warning is displayed, return to the program editing screen with soft key [CANCEL] and press soft key [GUIDANCE TWP] again, or eliminate the cause of the warning and retry the operation.

Warning	Description
"PROGRAM WRITE FAILED"	• The guidance screen was displayed from the foreground editing screen or MDI editing screen, and a block insertion or replacement operation was performed when the CNC was in the reset state or emergency stop state.
"PROGRAM CANNOT BE WRITTEN"	• A block insertion or replacement operation was performed after a program to be edited was updated due to program downloading by an external application while the guidance screen was displayed. • A block insertion or replacement operation was performed after the start of the main program was located by a reset while the guidance screen was displayed.
"WRITE PROTECT"	• A block insertion or replacement operation was performed when the editing or display was prohibited for a program to be edited. • A block insertion or replacement operation was performed when the memory protection signal (KEY signal) for program registration or editing was off. • A block insertion or replacement operation was performed when the operation level of the 8-level data protection function was lower than the protection level of a part program editing operation.

Display of the command type selection screen

When soft key [COMMAND TYPE] is pressed, the command type selection screen is displayed. When the command type is changed on the command type selection screen, the values set on the tilted working plane data setting screen are cleared.
When a tilted working plane indexing is included in the block at the cursor position on the program editing screen, you can display the guidance screen, change the command type on the command type selection screen, and then perform block replacement. In this case, the block at the cursor position on the program editing screen is replaced with a block created from the command type and command data that have been set.

12.2.9.3 Details of the tilted working plane data setting screen

The following six tilted working plane indexing are supported.
For details of the commands, see II-22.3, "TILTED WORKING PLANE INDEXING".

- G68.2 / G68.4 (Euler's angle)
- G68.2 / G68.4 P1 (Roll-Pitch-Yaw angle)
- G68.2 / G68.4 P2 (3 points specification)
- G68.2 / G68.4 P3 (2 vectors specification)
- G68.2 / G68.4 P4 (Projection angle)
- G68.3 (Tool Axis Direction)

G68.2 is an absolute command, and G68.4 is an incremental command.

G68.2 / G68.4(Euler's angle)

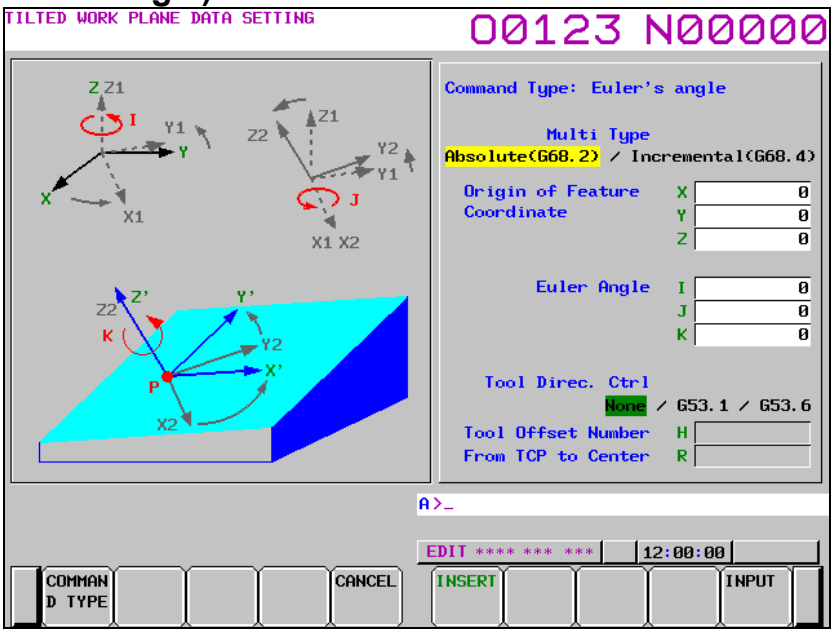


Fig. 12.2.9.3 (a) Tilted working plane data setting screen-Euler's angle(10.4-inch display unit)

- Multi Type

  Absolute:

  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.

  Incremental:

  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate

  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Euler's angle

  I : Specify an angle of rotation around the Z-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).
  This rotation determines coordinate system 1 (X1-Y1-Z1) from the workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type) (X-Y-Z).

  J : Specify an angle of rotation around the X-axis of coordinate system 1.
  This rotation determines coordinate system 2 (X2-Y2-Z2) from coordinate system 1 (X1-Y1-Z1).

  K : Specify an angle of rotation around the Z-axis of coordinate system 2.
  After this rotation, a feature coordinate system is obtained by shifting the origin of the workpiece coordinate system by the coordinates specified in "Origin of Feature Coordinate".

G68.2 / G68.4(Roll-Pitch-Yaw angle)

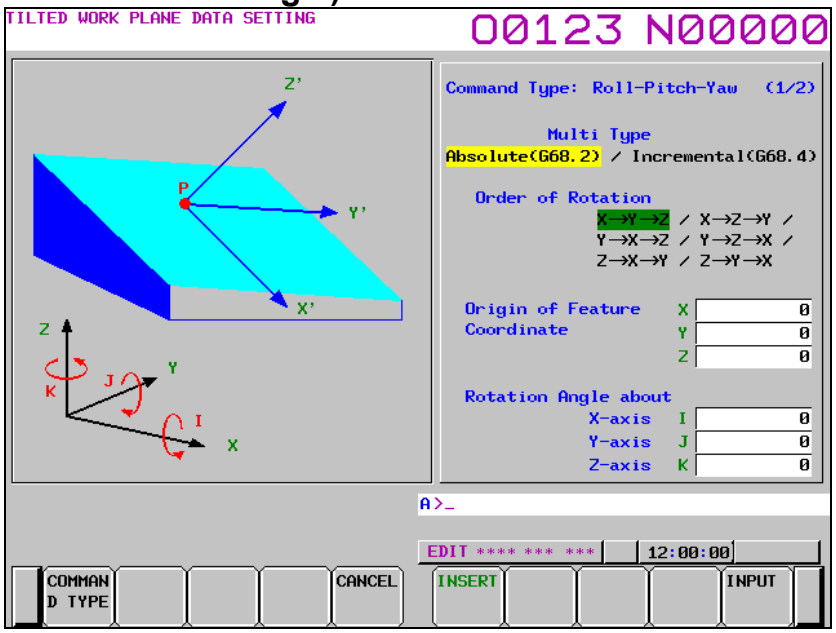


Fig. 12.2.9.3 (b) Tilted working plane data setting screen- Roll-Pitch-Yaw angle(10.4-inch display unit)

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Order of Rotation
  Select an order in which the X-axis, Y-axis, and Z-axis are rotated in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). The selectable rotation orders are as follows:

	1st rotation axis	2nd rotation axis	3rd rotation axis
X→Y→Z	X-axis	Y-axis	Z-axis
X→Z→Y	X-axis	Z-axis	Y-axis
Y→X→Z	Y-axis	X-axis	Z-axis
Y→Z→X	Y-axis	Z-axis	X-axis
Z→X→Y	Z-axis	X-axis	Y-axis
Z→Y→X	Z-axis	Y-axis	X-axis

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about X-axis
  Specify an angle of rotation around the X-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about Y-axis
  Specify an angle of rotation around the Y-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about Z-axis
  Specify an angle of rotation around the Z-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

G68.2 / G68.4(3 points specification)

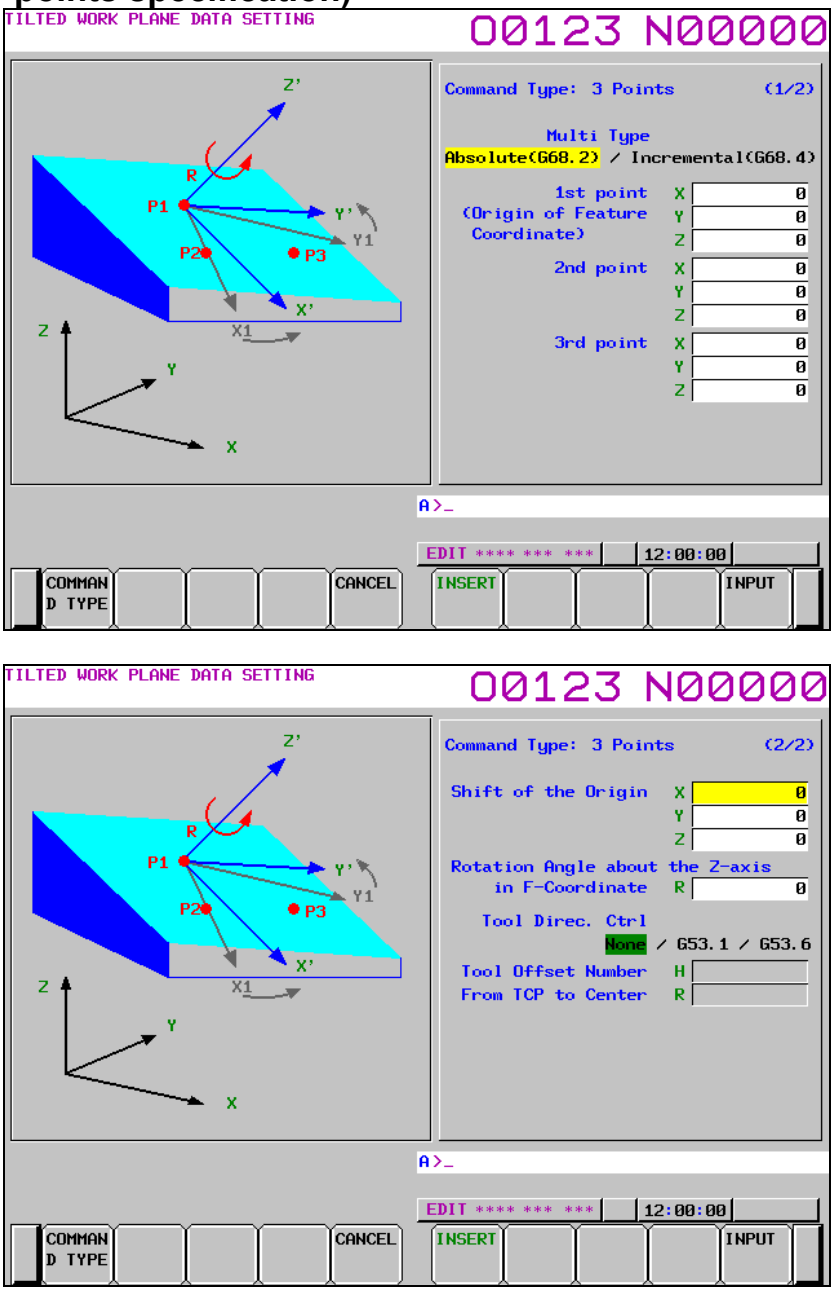


Fig. 12.2.9.3 (c) Tilted working plane data setting screen- 3 points specification(10.4-inch display unit)

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Shift of the Origin
  Specify, in a feature coordinate system, an amount of shift from the feature coordinate system origin specified for the 1st point (point P1).

- Rotation Angle about the Z-axis in F-Coordinate
  Specify an angle of rotation around the Z-axis of a feature coordinate system. The direction of rotation angle R is positive when a rotation is made clockwise as viewed in the Z-axis direction of the feature coordinate system.

- 1st point (Origin of Feature Coordinate)
  Specify the origin (X, Y, and Z of point P1) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- 2nd point
  Specify the coordinates (X, Y, and Z of point P2) of the 2nd point as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). The 1st point and 2nd point determine the X-axis of the feature coordinate system.

- 3rd point
  Specify the coordinates (X, Y, and Z of point P3) of the 3rd point as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). Among the directions normal to the X-axis, a direction with a smaller angle relative to the P1 → P3 vector is the Y-axis of the feature coordinate system.

G68.2 / G68.4(2 vectors specification)

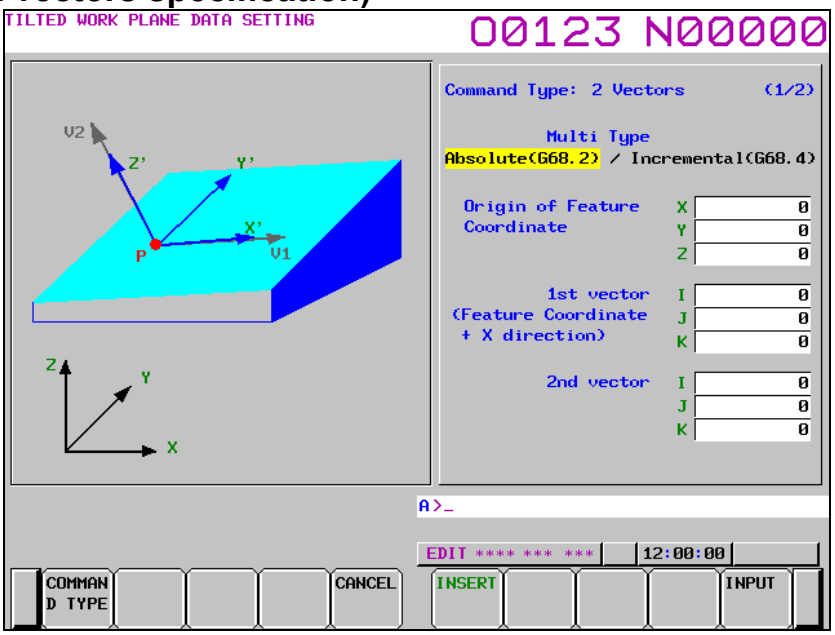


Fig. 12.2.9.3 (d) Tilted working plane data setting screen-2 vectors specification(10.4-inch display unit)

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- V1 (X') vector
  Specify the X-axis direction vector of a feature coordinate system as values in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- V2 (Z') vector
  Specify the Z-axis direction vector of a feature coordinate system as values in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

G68.2 / G68.4(Projection angle)

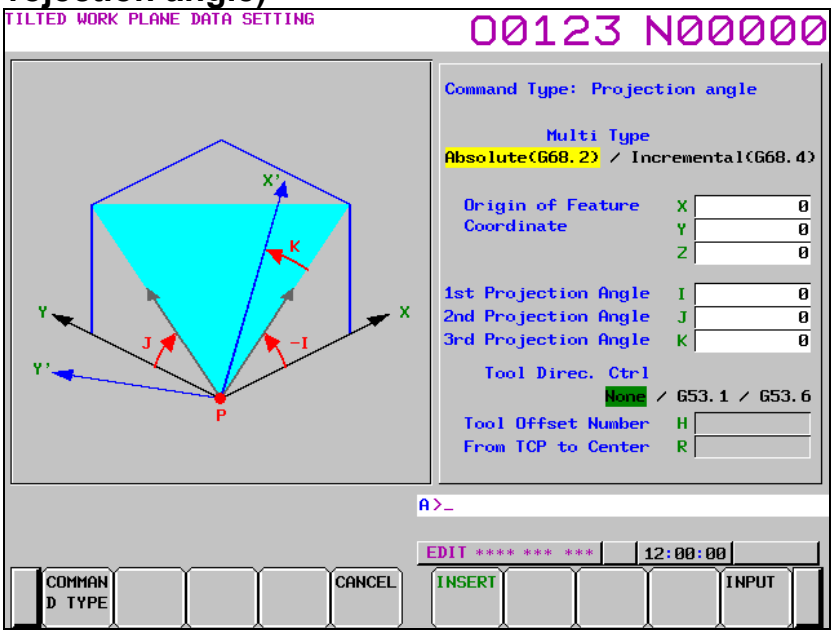


Fig. 12.2.9.3 (e) Tilted working plane data setting screen-Projection angle(10.4-inch display unit)

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Projection angle
  I : Specify a projection angle relative to the X-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).
  J : Specify a projection angle relative to the Y-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

K : Specify an angle of rotation around the Z-axis of a feature coordinate system.

G68.3(Tool Axis Direction)

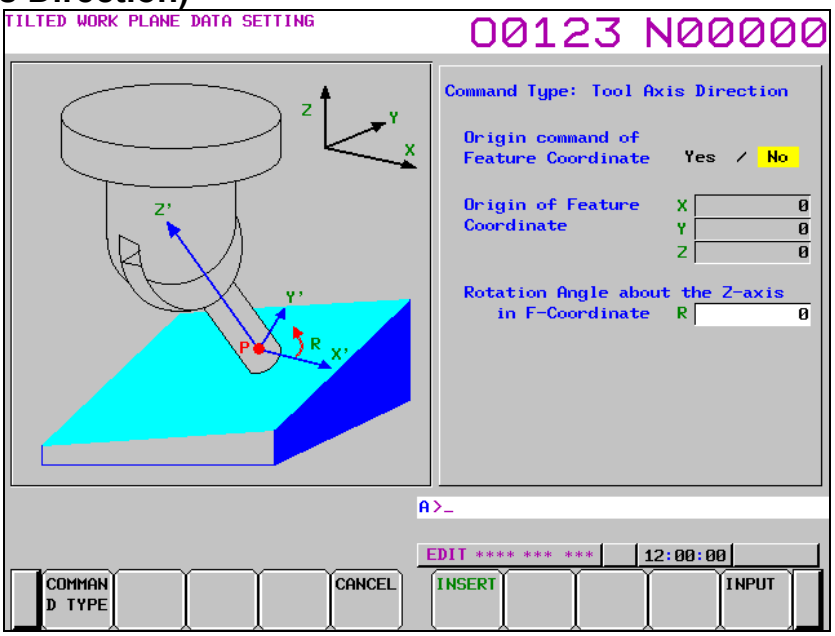


Fig. 12.2.9.3 (f) Tilted working plane data setting screen-Tool Axis Direction(10.4-inch display unit)
(When "No" is selected in "Origin command of Feature Coordinate")

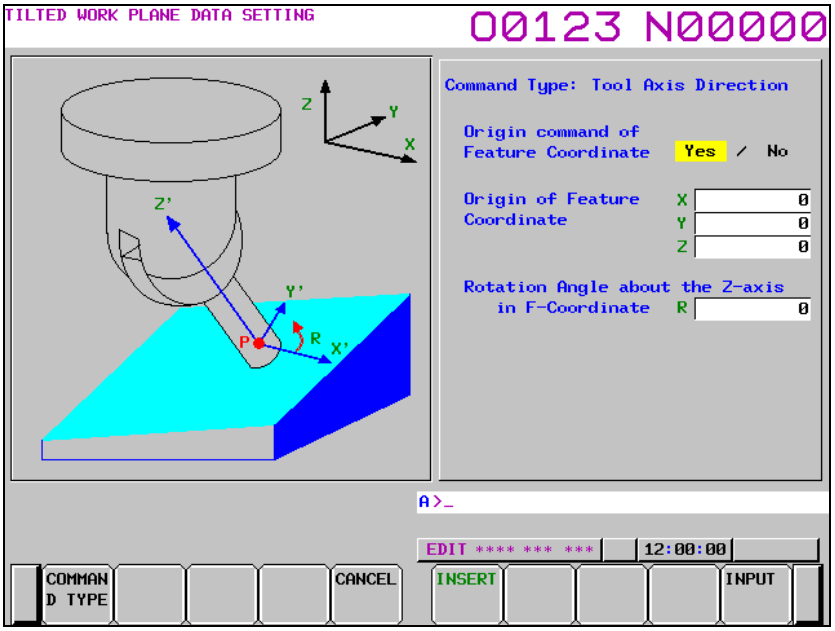


Fig. 12.2.9.3 (g) Tilted working plane data setting screen-Tool Axis Direction(10.4-inch display unit)
(When "Yes" is selected in "Origin command of Feature Coordinate")

- Origin command of Feature Coordinate
  Select whether to specify a feature coordinate system origin.
  Yes : A feature coordinate system origin is specified.
  No : A feature coordinate system origin is not specified.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system.
  This setting cannot be made if "No" is selected in "Origin command of Feature Coordinate".

- Rotation Angle about the Z-axis in F-Coordinate
  Specify an angle of rotation around the Z-axis of a feature coordinate system. The direction of rotation angle R is positive when a rotation is made clockwise as viewed in the Z-axis direction of the feature coordinate system.

12.2.9.4 Limitation

This function is supported on 10.4-inch, 15-inch, and 19-inch display units. This function is not displayed on 8.4-inch display unit.

Screens of a 15/19-inch display unit

This section describes the screens displayed by pressing function key 

. The screens include a program editing screen, program folder list display screen, and screens for displaying the command states of the program currently being executed.

1. Program screen
2. Program folder screen
3. Next block display screen
4. Program check screen

On the program screen, you edit the program that is currently selected, and display the block that is currently executed during program operation. In MDI mode, you also edit an MDI operation program, and display the block that is currently executed.

12.2.10 Program Contents Display (15/19-inch Display Unit)

Displays the program currently being executed in MEM mode.

Displaying the program being executed

Procedure

1 Press function key 

to display the program screen.
2 Press vertical soft key [PROGRAM].
The cursor is positioned at the block currently being executed.

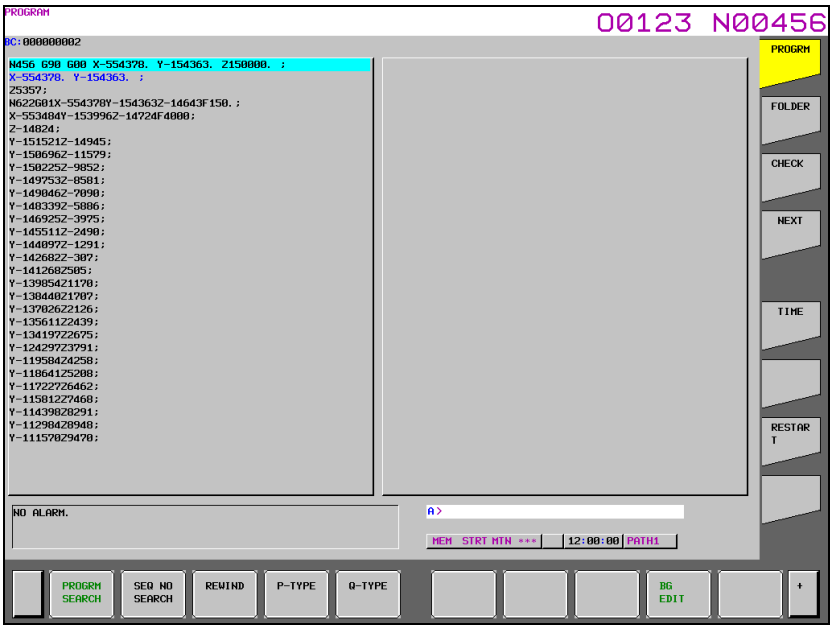


Fig. 12.2.10 (a) Screen for displaying the program being executed (15-inch display unit)

12.2.10.1 Displaying the executed block

For an explanation of how to display the executed block, see Subsection 12.2.1.1, "Displaying the executed block".

12.2.11 Editing a Program (15/19-inch Display Unit)

A program can be edited in the EDIT mode.
Two modes of editing are available. One mode is word editing, which performs word-by-word editing. The other is character editing, which performs character-by-character editing.
For program creation and editing operation, see Chapter III-9, “CREATING PROGRAMS “ and Chapter III-10, “EDITING PROGRAMS”.

Displaying the program editing screen

Procedure

1 Press function key 

to display the program screen.

2 Press vertical soft key [PROGRAM].

- Word editing

Editing operations such as text insertion, modification, and deletion, and cursor movements are performed on a word-by-word basis.

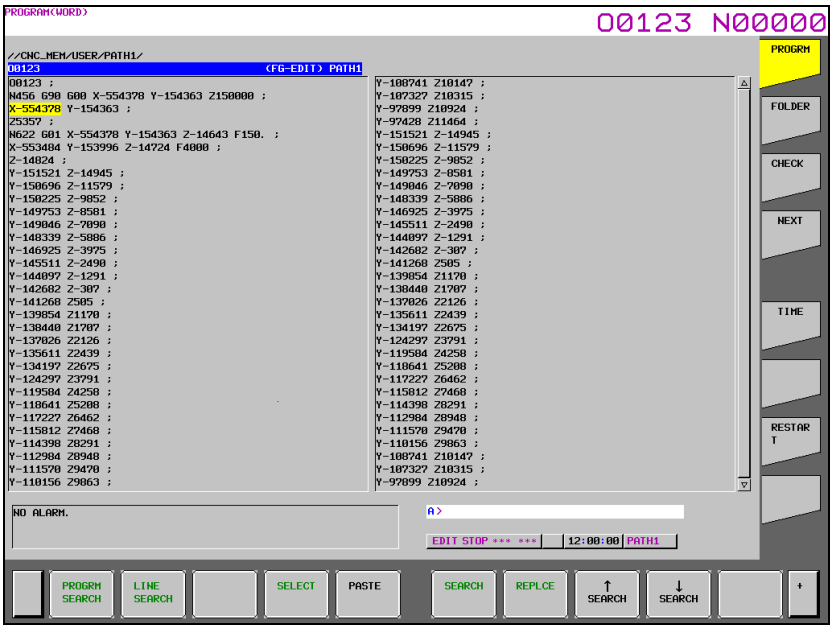


Fig. 12.2.11 (a) Program word editing screen (15-inch display unit)

- Character editing

Program editing operations and cursor movements are performed on a character-by-character basis as with a general text editor.
Text is input directly to the cursor position instead of using the key input buffer.

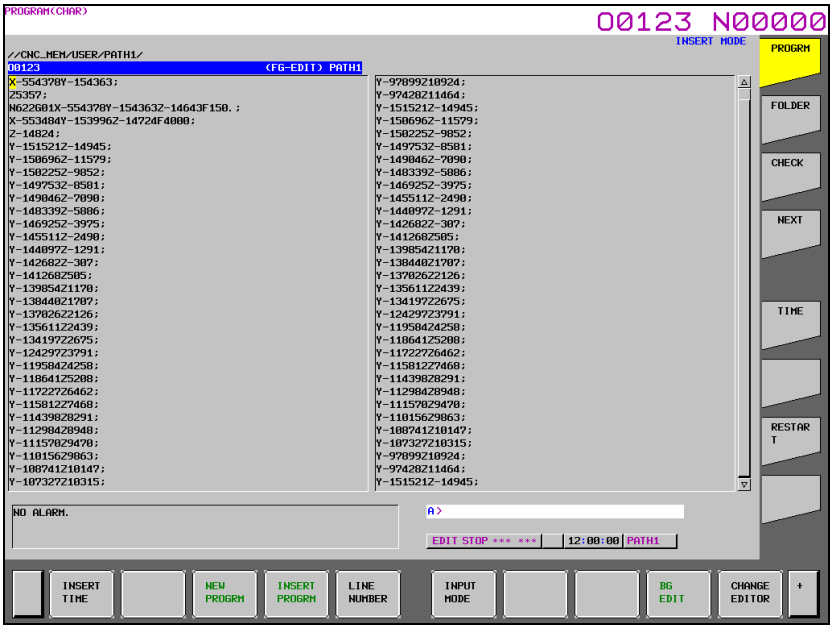


Fig. 12.2.11 (b) Program character editing screen (15-inch display unit)

Switching between program editing modes

You can switch between word editing and character editing with horizontal soft keys.

Procedure

1 Press function key  to display the program screen.
2 Press vertical soft key [PROGRAM].
3 Press continuous menu key  to display horizontal soft key [CHANGE EDITOR].

4 Pressing horizontal soft key [CHANGE EDITOR] switches the editing mode between word editing and character editing.

12.2.12 Program Screen for MDI Operation (15/19-inch Display Unit)

During MDI operation or editing of an MDI operation program in the MDI mode, the program currently being executed mode is displayed.
For MDI operation, see Section III-4.2, “MDI Operation”.

Procedure for displaying the program screen for MDI operation

Procedure

1 Press function key 

.

2 Press vertical soft key [PROGRAM].
The program input from the MDI is displayed.

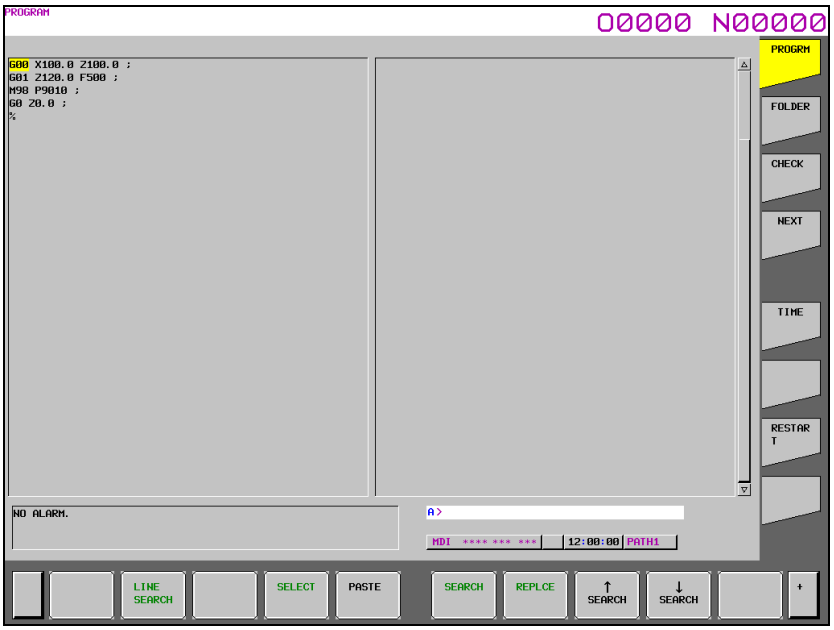


Fig. 12.2.12 (a) MDI operation program screen (15-inch display unit)

12.2.13 Program Folder Screen (15/19-inch Display Unit)

A list of programs registered in the program memory is displayed.
For the program folder screen, see Chapter, “PROGRAM MANAGEMENT”.

Displaying the program folder screen

Procedure

1 Press function key 

.

2 Press vertical soft key [FOLDER].

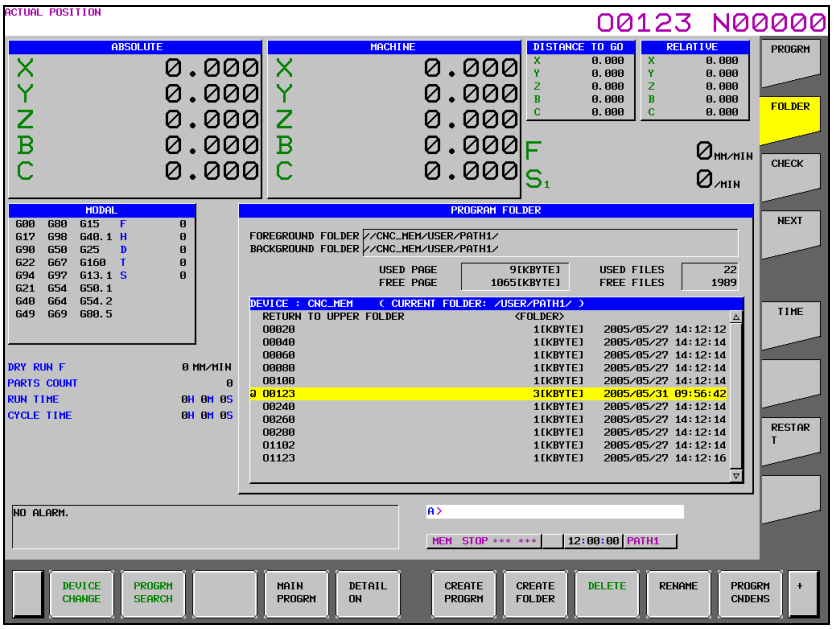


Fig. 12.2.13 (a) Program folder screen (15-inch display unit)

12.2.13.1 Split display on the program folder screen

Overview

On the program folder screen, the folder information display can be split into two folder information views, upper and lower, as shown in the Fig. 12.2.13.1 (a).

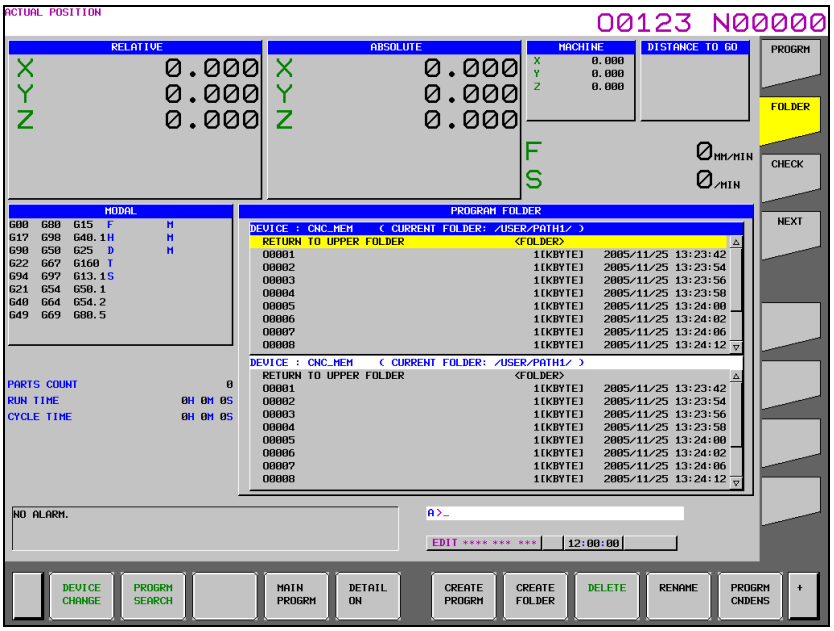
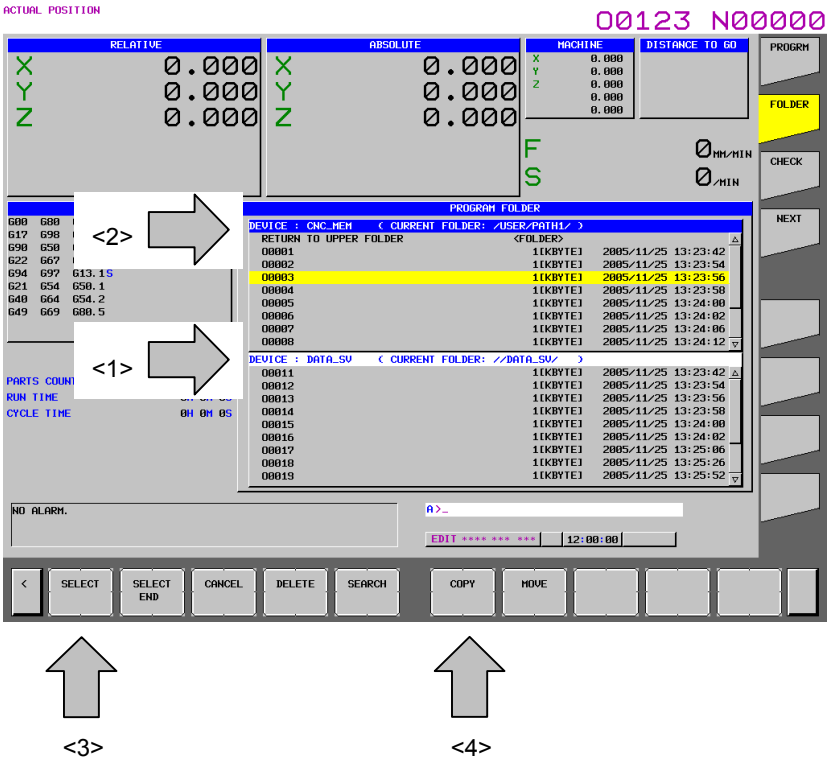


Fig. 12.2.13.1 (a) Program folder screen (split-display screen)(15-inch display unit)

On the split folder display, file information about different sets of devices or folders can be shown in different views at the same time, so you can copy or move files easily.

Example of copying a file from the CNC_MEM to the Data Server



Procedure

<1> Change to a destination folder on the Data Server.
<2> Change to a folder on the CNC_MEM that contains a file you want to copy.
<3> Select the file.
<4> Copy the file.

Procedure for switching to split display

Procedure

1 Press function key PROG to display the program folder screen.
2 Press horizontal soft key [(OPRT)].
3 Press continuous menu key twice.
Horizontal soft key [MULTI LIST] appears.

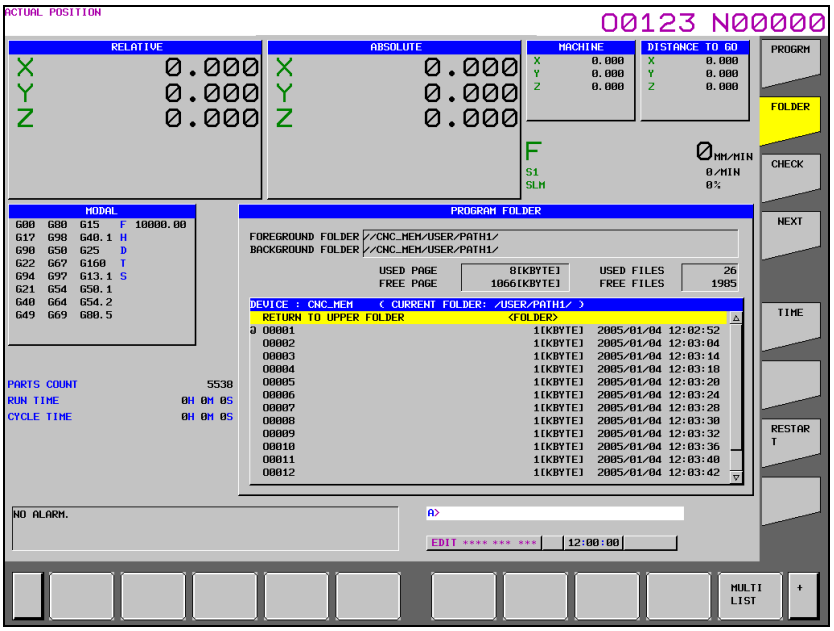


Fig. 12.2.13.1 (b) Program folder screen normal screen)(15-inch display unit)

4 Press horizontal soft key [MULTI LIST].
The folder information display is split into two folder views, upper and lower, which show the same folder information. Immediately after the splitting, the upper folder view becomes active for operations. At the same time, horizontal soft key [SINGLE LIST] appears.

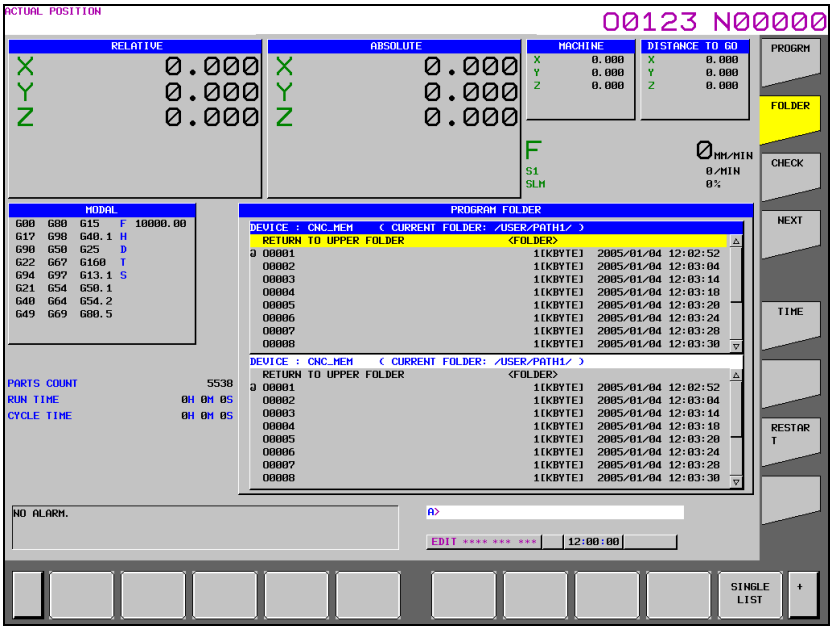


Fig. 12.2.13.1 (c) Program folder screen (split-display screen)(15-inch display unit)

5 Pressing horizontal soft key [SINGLE LIST] takes you back to the original program folder display. Then, the folders are shown which have been selected in the active view for operations on the split display.

Procedure for switching between active folder views for file operations on the split display

On the split folder display, you can switch between active folder views for file operations as described below.
In the active folder view for file operations, the cursor is displayed on the file list.

Procedure

1 When the upper folder view is active for file operations and you press MDI key [SHIFT] and cursor key [↓], the lower folder view becomes active for file operations.

2 When the lower folder view is active for file operations and you press MDI key [SHIFT] and cursor key [↑], the upper folder view becomes active for file operations.

NOTE

1 In step 1, even if you press MDI key [SHIFT] and cursor key [↑] when the upper folder view is active for file operations, the lower folder view becomes active for file operations.

2 In step 2, even if you press MDI key [SHIFT] and cursor key [↓] when the lower folder view is active for file operations, the upper folder view becomes active for file operations.

Explanation

- File operations on the split display

The following operations are allowed on the split display.
<1> You can select and display devices or folders individually in each folder view.
<2> You can copy or move files between folders selected in different folder views.
(For details, see the section on operations of copying/moving between devices.)
In the selected folder view, you can perform the same operations as on the normal screen.

- Path switching

When path switching is performed while the split display is provided for a multi-path system, the folder view in which the CNC MEM is selected as a device is changed to a view of the last selected folder for each path on the CNC MEM.

- Device information display

The device information as shown in the normal program folder screen (portion in a dotted box in the Fig. 12.2.13.1 (d)) is not shown on the split display (Fig. 12.2.13.1 (c)).

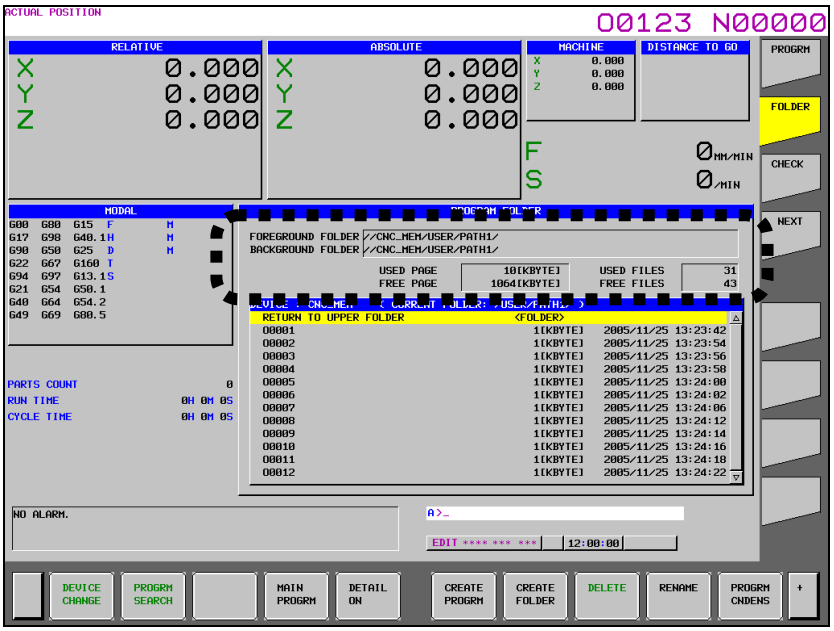


Fig. 12.2.13.1 (d) Program folder screen (normal screen)(15-inch display unit)

You should see the device information on the normal screen.

Limitation

- Device that can be selected in both views at the same time on the split display

The same device of any of the following types can be selected in both views at the same time on the split display.

- CNC MEM
- MEM CARD(binary format)
- Data Server

For other types, the same device cannot be selected for display in the folder views at the same time.

If any other type of device is selected when split display starts, the upper and lower folder views cannot show the same sets of folders due to the above limitation. Therefore, the lower folder view shows the folder selected as a foreground folder.

- Display units that support the split display

The split display is supported on 10.4-inch, 15-inch, and 19-inch display units.

12.2.13.2 Folder tree display (15/19-inch display unit)

This is same as the 10.4-inch display units. Refer to 12.2.4.2 Folder tree display.

12.2.14 Next Block Display Screen (15/19-inch Display Unit)

Displays the block currently being executed and the block to be executed next.

Procedure for displaying the next block display screen

Procedure

1 Press function key 

.

2 Press vertical soft key [NEXT].
The G codes, addresses, command values specified in the block currently being executed and the next block are displayed.

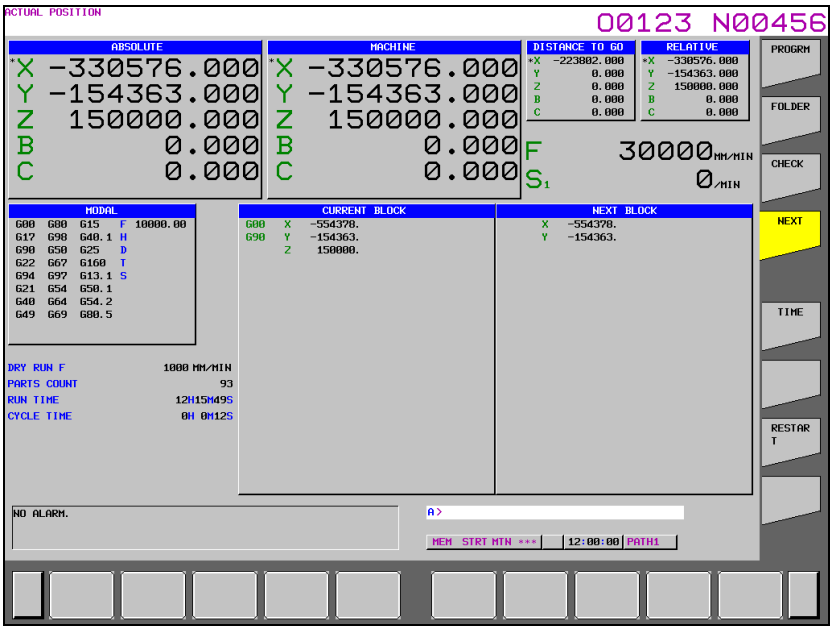


Fig. 12.2.14 (a) Next block display screen (15-inch display unit)

12.2.15 Program Check Screen (15/19-inch Display Unit)

Displays the program currently being executed, current position of the tool, and modal data.

Procedure for displaying the program check screen

Procedure

1 Press function key 

.

2 Press vertical soft key [CHECK].
The program currently being executed, current position of the tool, and modal data are displayed.

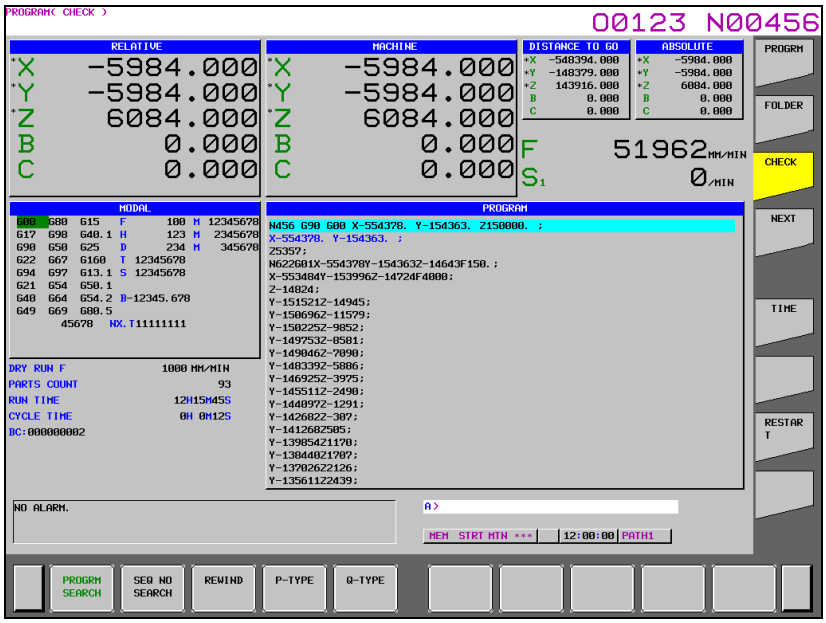


Fig. 12.2.15 (a) Program check screen (15-inch display unit)

Explanation

- Program display

The program currently being executed is displayed.
The block being executed is displayed in reverse video.

- Current position display

The current position in the relative coordinate system, workpiece coordinate system, and machine coordinate system, and the remaining amount of travel are displayed.

- Modal G codes

Up to 24 modal G codes are displayed.

12.2.16 Background Editing (15/19-inch Display Unit)

Editing one program during execution of another program is referred to as background editing. You can perform the same edit operations in the background as those in normal editing (foreground editing).

On a 15-inch or 19-inch display, you can perform background editing for the programs displayed in multiple tiled windows. You can switch from a program displayed in one of the tiled windows to another to perform copy, paste, and other edit operations, which allows you to edit programs efficiently.
You can edit four programs simultaneously.

Function

- Background editing

A program other than the currently selected program can be edited. Background editing can be performed in any mode.

- EDIT mode and reference mode

When background editing is started in the EDIT mode, the program can be edited. When background editing is started in the reference mode, the program is opened in the read-only mode and cannot be edited.
While a program opened in the EDIT mode cannot be run, a program opened in the reference mode can be run.

- Program selected in the foreground

If the program selected in the foreground is specified as a program to be edited in the background, background editing is started in the read-only mode. The text at an arbitrary position of the program currently being executed can be checked.

- Switching from the program directory screen to the editing screen

By selecting a program from the program folder screen, background editing can be started.
On the program folder screen, move the cursor to a program to be edited in the background, then press the MDI key 

. The screen display switches to the background editing screen.
Background editing can be started without entering a program name.

- Background editing on a multi-path system

Background editing does not depend on the path.
Even if the path to be operated is changed using the path selection signal, the background editing currently being executed is just continued.

NOTE
The following programs cannot be edited in the background. They can be opened only in the read-only mode.
- Running program
- Main program
- Program with the edit disable attribute

Display

When background editing starts, the ordinary program editing screen switches to the background editing screen.
When two or more programs are edited in the background, the screen is split to display these programs. For a 15-inch or 19-inch display unit, you can edit four programs simultaneously.

- Word editing

Fig. 12.2.16 (a) shows background word editing performed simultaneously for two programs (right and left programs). On the status line at the top of the window for each program, the program name and “BG-EDIT” (indicating background editing in progress) are displayed.
The status line of the program being edited is displayed in reverse video.

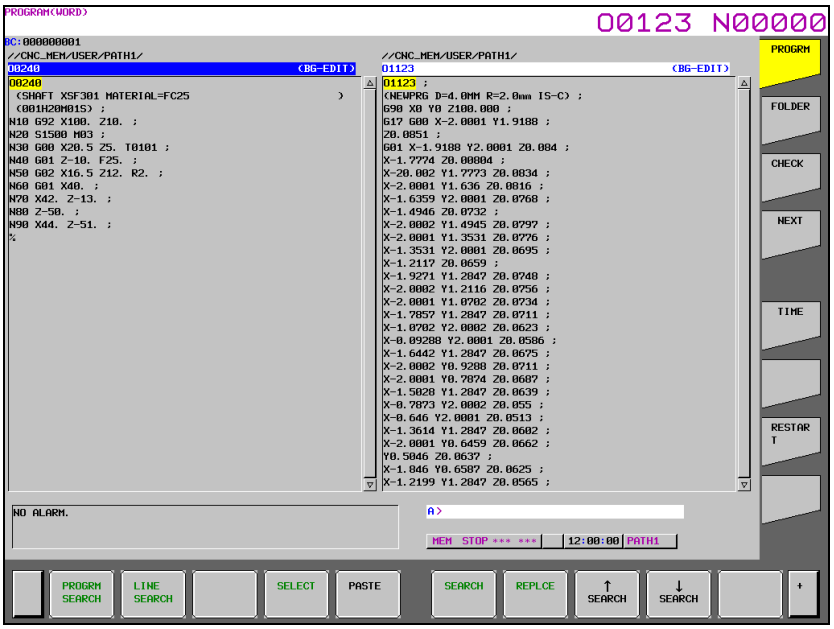


Fig. 12.2.16 (a) Background editing screen (word editing) (15-inch display unit)

- Character editing

Fig. 12.2.16 (b) shows background character editing performed simultaneously for two programs (right and left programs). Similarly to word editing, at the top of the window for each program, the status line is displayed. In addition, the current input mode (INSERT MODE or OVERWRITE MODE) is displayed at the upper right of the screen for character editing.
The status line of the program being edited is displayed in reverse video.

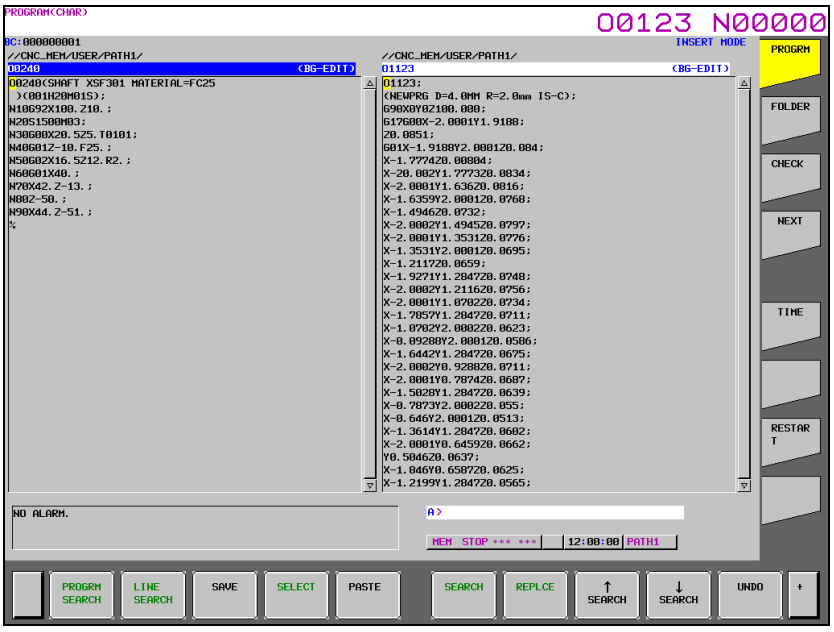


Fig. 12.2.16 (b) Background editing screen (character editing) (15-inch display unit)

- Editing status

The following items are displayed on the status line and program editing area according to the background editing status.

Editing status	Displayed items
No program selected	(BG-EDIT) “NO PROGRAM” is displayed in the editing area.

Editing status	Displayed items
Program opened	program-name + (BG-EDIT)
Read-only program opened	program-name + (BG:READ ONLY) The contents of the program are displayed in green.

Starting background editing from the editing screen

Procedure (word editing)

Method 1

1 Press function key .
2 Press soft key [PROGRAM].
3 Enter a program name.
4 Press soft key [BG EDIT].

NOTE
1 Background editing is performed in the EDIT mode.
2 If a nonexistent program is specified, a new program is created.

Method 2

1 Press function key .
2 Press soft key [PROGRAM].
3 Press soft key [BG EDIT].
4 Enter a program name.
5 Press soft key [EDIT EXEC] to start background editing in the EDIT mode or soft key [REF EXEC] to start background editing in the reference mode.

NOTE
If a nonexistent program is specified,
- Pressing soft key [EDIT EXEC] causes a new program to be created.
- Pressing soft key [REF EXEC] causes the warning message “SPECIFIED PROGRAM NOT FOUND” to appear. A new program is not created.

Method 3

1 Press function key .
2 Press soft key [PROGRAM].
3 Press soft key [BG EDIT].
4 Press soft key [EDIT EXEC] to start background editing in the EDIT mode or soft key [REF EXEC] to start background editing in the reference mode. At this time, “NO PROGRAM” is displayed on the screen.
5 Enter a program name.
6 Press soft key [PROGRM SEARCH].

NOTE
If a nonexistent program is specified, the warning message “SPECIFIED PROGRAM NOT FOUND” is displayed. A new program is not created.

Procedure (character editing)

Method 1

1 Press function key .

2 Press soft key [PROGRAM].
3 Press soft key [BG EDIT].
4 Enter a program name.
5 Press soft key [EDIT EXEC] to start background editing in the EDIT mode or soft key [REF EXEC] to start background editing in the reference mode.

NOTE
If a nonexistent program is specified,
- Pressing soft key [EDIT EXEC] causes a new program to be created.
- Pressing soft key [REF EXEC] causes the warning message “SPECIFIED PROGRAM NOT FOUND” to appear. A new program is not created.

Method 2

1 Press function key 

.
2 Press soft key [PROGRAM].
3 Press soft key [BG EDIT].
4 Press soft key [EDIT EXEC] to start background editing in the EDIT mode or soft key [REF EXEC] to start background editing in the reference mode. At this time, “NO PROGRAM” is displayed on the screen.
5 Press soft key [PROGRM SEARCH].
6 Enter a program name.
7 Press soft key [PROGRM NAME].

NOTE
If a nonexistent program is specified, the warning message “SPECIFIED PROGRAM NOT FOUND” is displayed. A new program is not created.

Starting background editing from the program folder screen

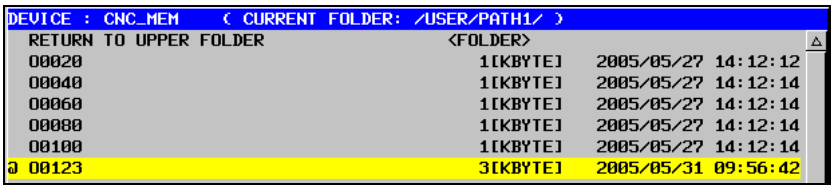
By selecting a program from the program folder screen, background editing can be started. The cursor is used to select a program. You do not need to enter a program name.

Procedure

1 Press function key 

.
2 Press vertical soft key [FOLDER].
3 Move the cursor to a program to be edited.
4 Press the MDI key 

. The screen display is switched to the background editing screen.



NOTE
When background editing is started from the program folder screen, the EDIT mode is set.
For the following programs, the reference mode is set, however:
- Running program
- Main program
- Program with the edit disable attribute

Background editing operation

- Editing operation
The same editing operations as performed in the foreground can be performed.

- Switching from a program to another for editing
To switch from a program to another for editing when editing multiple programs simultaneously in the background, press 

key, then  or .

- Editing operation in the read-only mode
Cursor movement and page switching are possible. However, program editing is disabled.

- Copying and pasting text between programs
Text in a program selected and copied during program editing is kept held after the program to be edited is changed. You can paste text in a program onto another program.

Background editing end operation

Background editing can be ended using the procedure described below.
The procedure for ending background editing of one program and that for ending all background editing of multiple programs are shown below.

- Ending editing of one program
1 Press function key 

.
2 Press vertical soft key [PROGRAM].
3 Select a program for which to end editing.
4 Press horizontal soft key [BG END].

- Ending editing of all programs
1 Press function key 

.
2 Press vertical soft key [PROGRAM].
3 Press horizontal soft key [BG ALL END].

To return to ordinary foreground editing, end all background editing. If at least one program is being edited in the background, you cannot return to foreground editing.

12.2.17 Stamping the Machining Time (15/19-inch Display Unit)

The execution times of the most recently executed ten programs can be displayed in hours, minutes, and seconds.
The calculated machining time can be inserted as a comment of the program to check the machining time on the program folder screen.

Procedure for stamping the machining time

Procedure

- Displaying the machining time

1 Press function key PROG.

2 Press vertical soft key [TIME]. The machining time display screen appears.

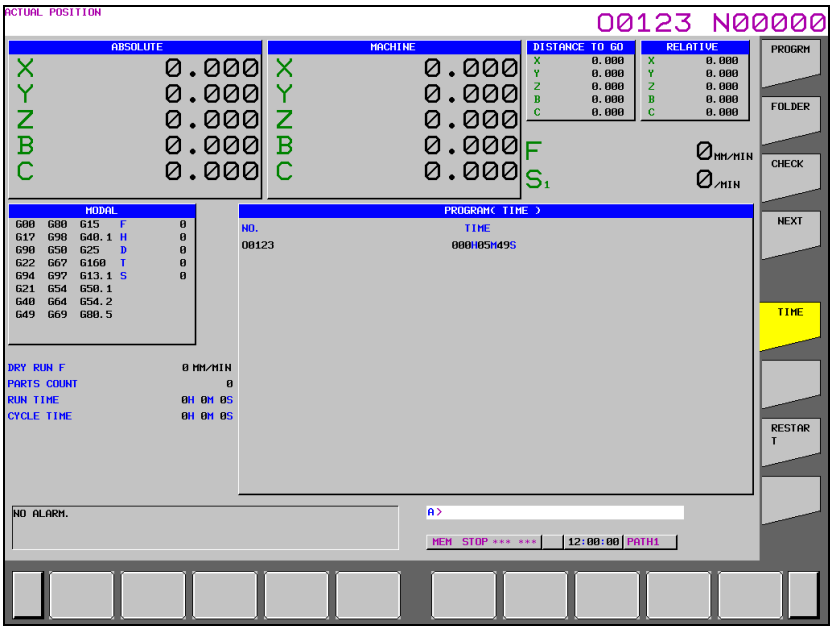


Fig. 12.2.17 (a) Machining time display screen (15-inch display unit)

- Calculating the machining time

1 Select the memory operation mode, then press the RESET key.

2 Select the program screen, then select a program of which machining time you want to calculate.

3 Execute the program to perform actual machining.

4 When the RESET key is pressed or, M02 or M30 is executed, the machining time count operation stops. When the machining time display screen is selected, the program number of the stopped main program and its machining time are displayed.

5 The Fig. 12.2.17 (b) shows the screen when the machining times of the ten main programs O0020, O0040, …, and O0200 are displayed and the screen when the machining time of O0220 is newly calculated after that.

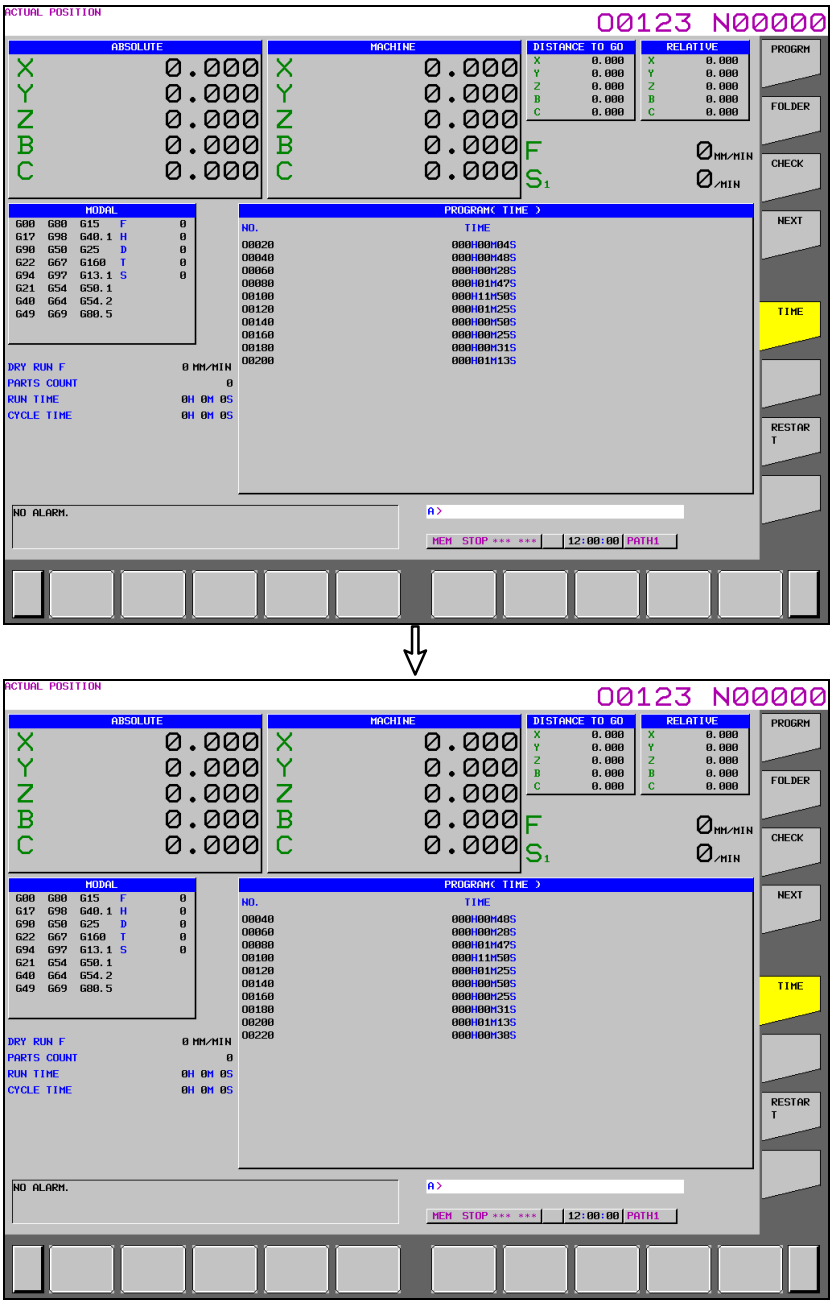


Fig. 12.2.17 (b) Stamping the machining time (15-inch display unit)

Procedure for inserting the machining time on the program screen

Procedure

You can display the machining time of a program as a comment of the program. The procedure is shown below:

1 To insert the calculated machining time of a program as a comment, the machining time of the program must be displayed on the machining time display screen. Before stamping the machining time of the program, check that the program number is displayed on the machining time display screen.

2 Set the edit mode or background editing state and select the program screen. Then, select a program of which machining time you want to insert.

3 For example, the machining time of O0100 is displayed on the machining time display screen. Press continuous menu key ▷ until horizontal soft key [INSERT TIME] appears. When horizontal soft key [INSERT TIME] is pressed, the start of the program is located, and the machining time of the program is inserted after the program name.



Fig. 12.2.17 (c) Program screen (15-inch display unit)

4 If a comment is written in the block containing the program number of a program of which machining time is to be inserted, the machining time is inserted after the comment.



Fig. 12.2.17 (d) Program screen (15-inch display unit)

Display on the program folder screen

The machining time of a program inserted in the program as a comment is displayed after the existing comment of the program on the program folder screen.

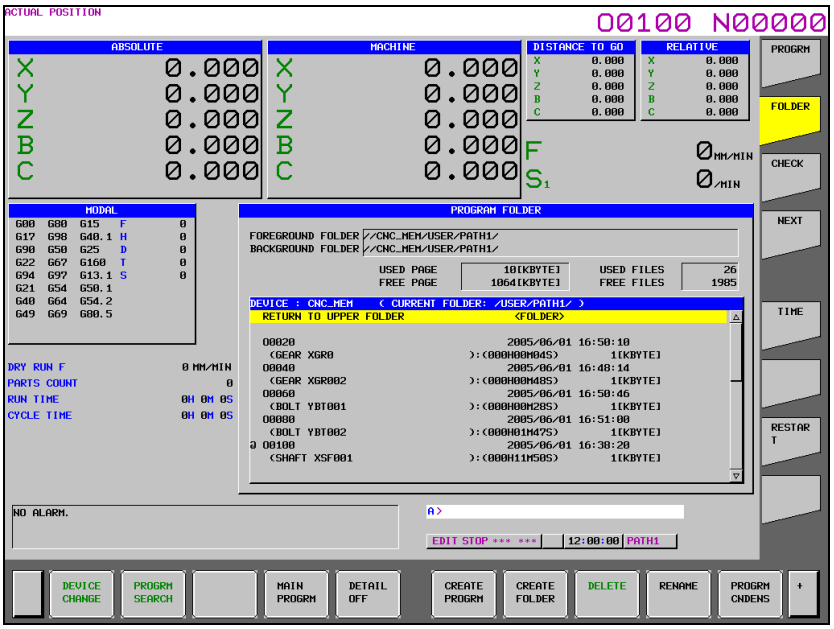


Fig. 12.2.17 (e) Program folder screen (15-inch display unit)

Explanation

- Machining time

The machining time is counted from the initial start after a reset in the memory operation mode to the next reset. If a reset is not performed during operation, the machining time is counted from the start to M02 or M30. The time during which operation is held is not counted, but the time used to wait for the completion of the M, S, T, and/or B functions is counted, however.

- Stamping the machining time

You can insert (stamp) the displayed machining time in a program stored in memory as a comment. The machining time is inserted as a comment after the program number.

- Program folder

You can display the machining time inserted after a program number on the program folder screen. This operation lets you know the machining time of each program to use the time as useful reference data when planning processes on site.

Limitation

- Alarm

The execution of a program may be held by an alarm during the machining time count operation. In this case, the machining time is counted until the alarm is released by a reset.

- M02

It may be specified that M02 does not reset the CNC, but returns completion signal FIN to the CNC to restart the program from the beginning successively (bit 5 (M02) of parameter No. 3404 is set to 0). In this case, when M02 returns completion signal FIN, the machining time count operation stops.

- Stamping the machining time

When an attempt is made to stamp the machining time of a program in the program, the machining time of the program may not be displayed on the machining time display screen. In this case, the machining time cannot be inserted in the program even if horizontal soft key [INSERT TIME] is pressed.

- Correcting the machining time

If an incorrect machining time is calculated (such as when a reset is made during the execution of a program), reexecute the program to calculate the correct machining time. The same program number may be displayed in two or more lines on the machining time display screen. In this case, the latest calculated machining time is inserted in the program.

- States of the stamped machining time

In the following states, the stamped machining time is displayed on the program folder screen as shown below.

1 When the comment of a program is longer than 16 characters
The 17th and subsequent characters of the comment are discarded and the machining time display field is left blank.

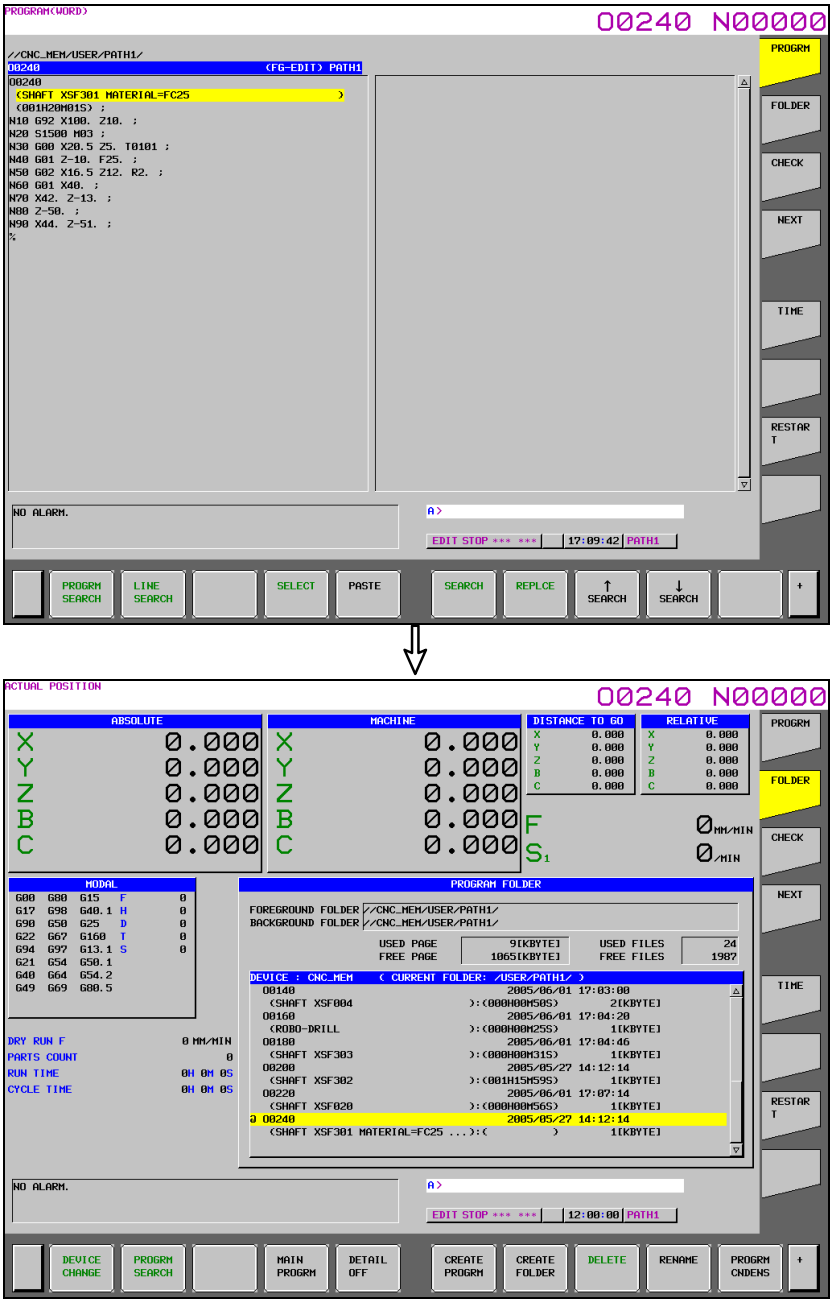


Fig. 12.2.17 (f) When the comment of a program is longer than 16 characters (15-inch display unit)

2 When two or more machining times are stamped
The first machining time is displayed.

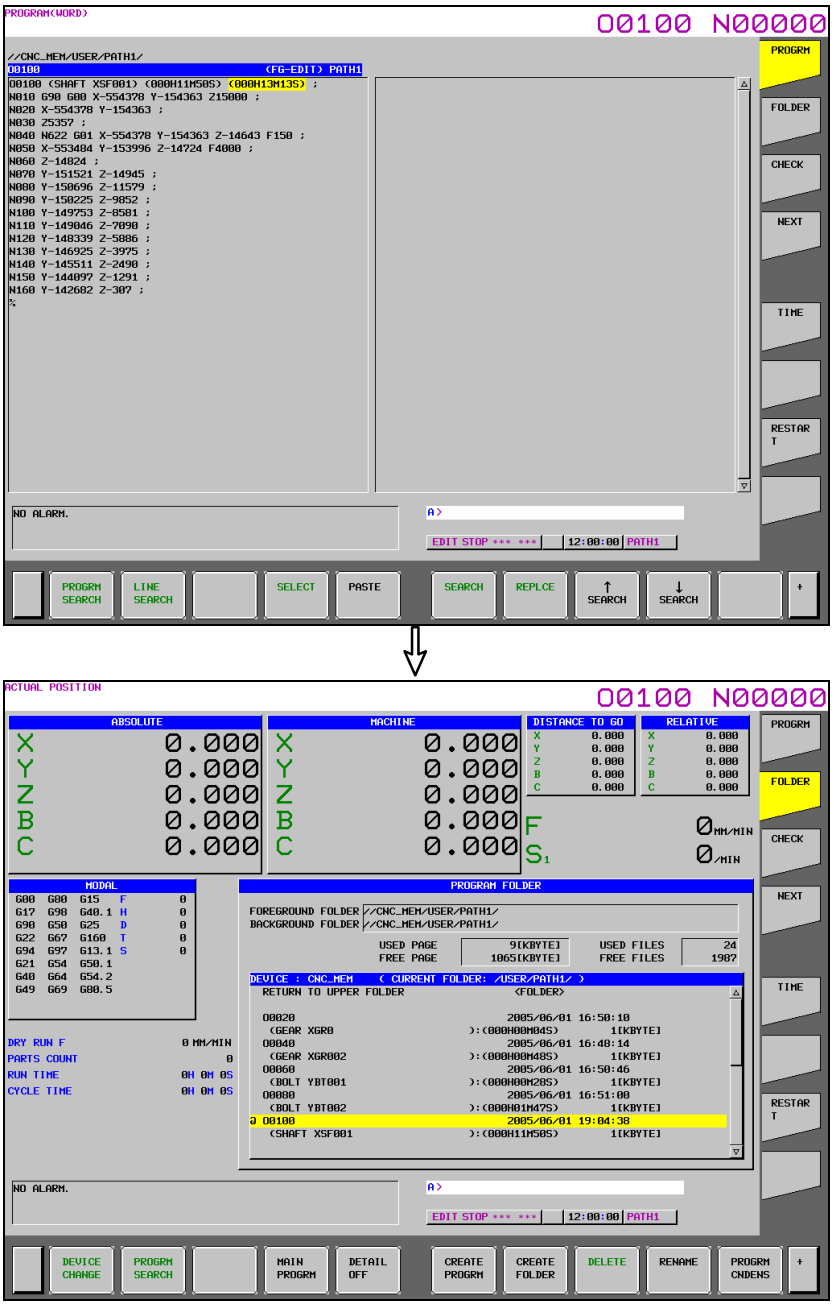


Fig. 12.2.17 (g) When two or more machining times are stamped (15-inch display unit)

3 When the format of an inserted machining time is not “hhhHmmMssS” (H following a 3-digit number, M following a 2-digit number, and S following a 2-digit number, in this order)
The machining time display field is left blank.

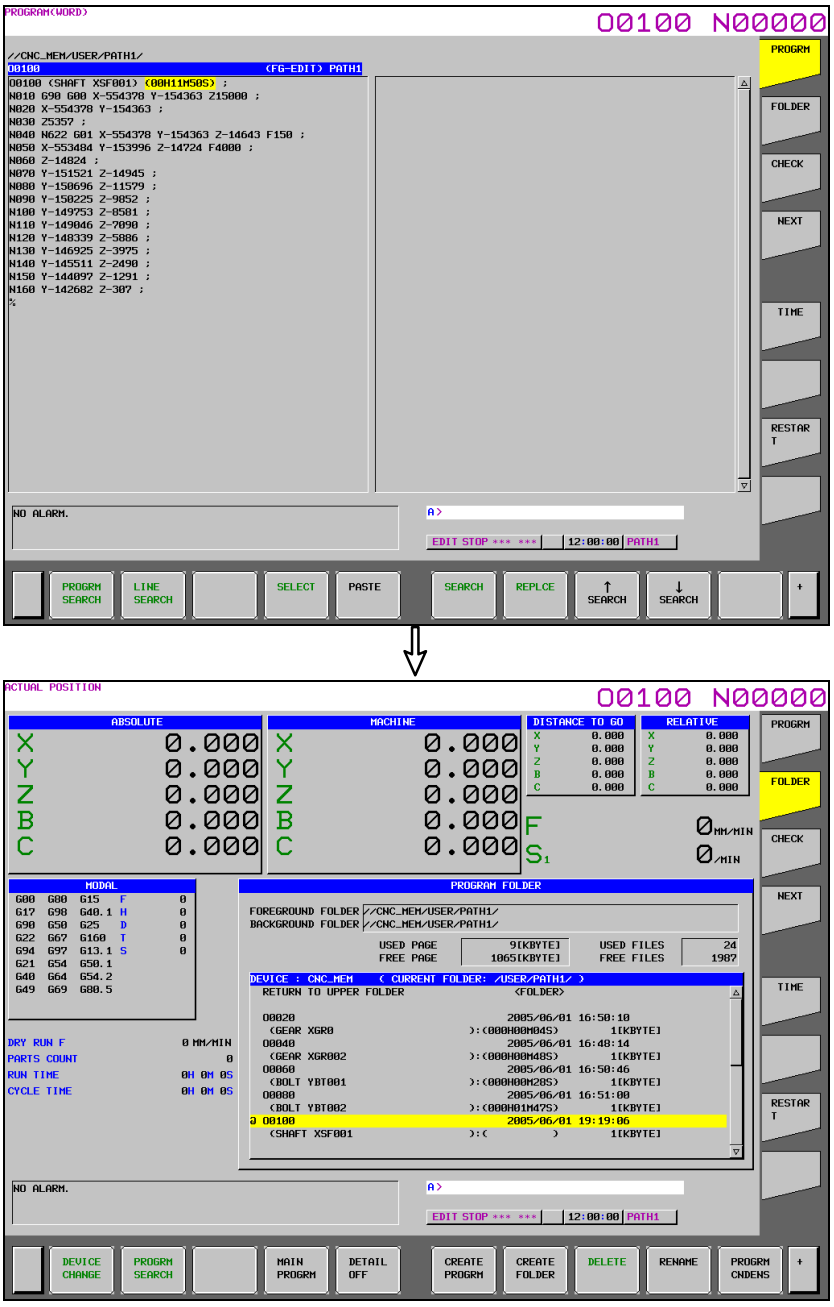


Fig. 12.2.17 (h) When the format of an inserted machining time is not “hhhHmmMssS” (H following a 3-digit number, M following a 2-digit number, and S following a 2-digit number, in this order) (15-inch display unit)

12.2.18 Screen for Assistance in Entering Tilted Working Plane Indexing (15/19-inch Display Unit)

The screens for assistance in entering tilted working plane indexing (hereinafter referred to as guidance screens) include a command type selection screen and a tilted working plane data setting screen. The command type selection screen is used to select a tilted working plane indexing. The tilted working plane data setting screen is used to set specified tilted working plane data required for the selected command. By making settings and performing operations on these guidance screens, a tilted working plane indexing block can be created.

The created block is reflected as a new insertion to a program being edited or as a modification to an existing block.
This function can be enabled by setting bit 1 (GGD) of parameter No. 11304 to 1.

Creation of a new block

The following describes the procedure for creating a tilted working plane indexing block on guidance screens and for inserting the block to a program being edited on a program editing screen.

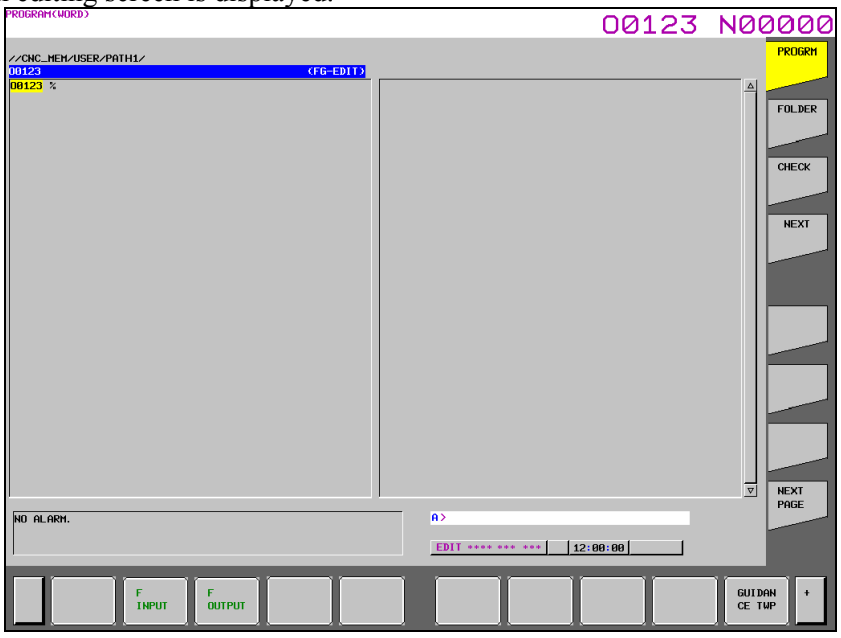
1 On a program editing screen, display a program to which you want to insert a tilted working plane indexing block.
The foreground editing screen, background editing screen, or MDI editing screen should be displayed.

- Displaying the foreground editing screen
  <1> Select the EDIT mode.
  <2> Press function key PROG.
  <3> Press vertical soft key [PROGRAM].

- Displaying the background editing screen
  <1> Press function key PROG.
  <2> Press soft key [FOLDER].
  <3> Press any of the cursor keys to move the cursor to a program to be edited in the background.
  <4> Press INPUT key.

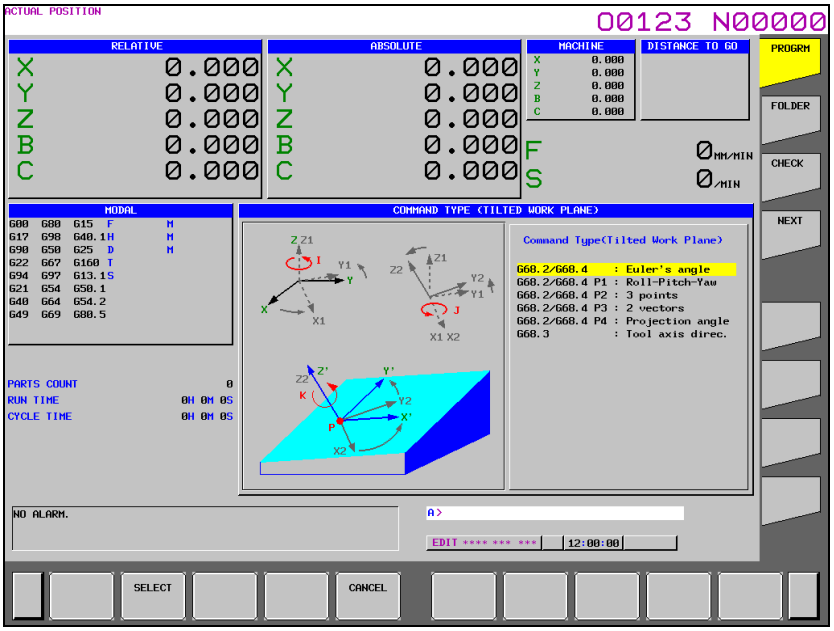
- Displaying the MDI editing screen
  <1> Select the MDI mode.
  <2> Press function key PROG.
  <3> Press vertical soft key [PROGRAM].

The program editing screen is displayed.

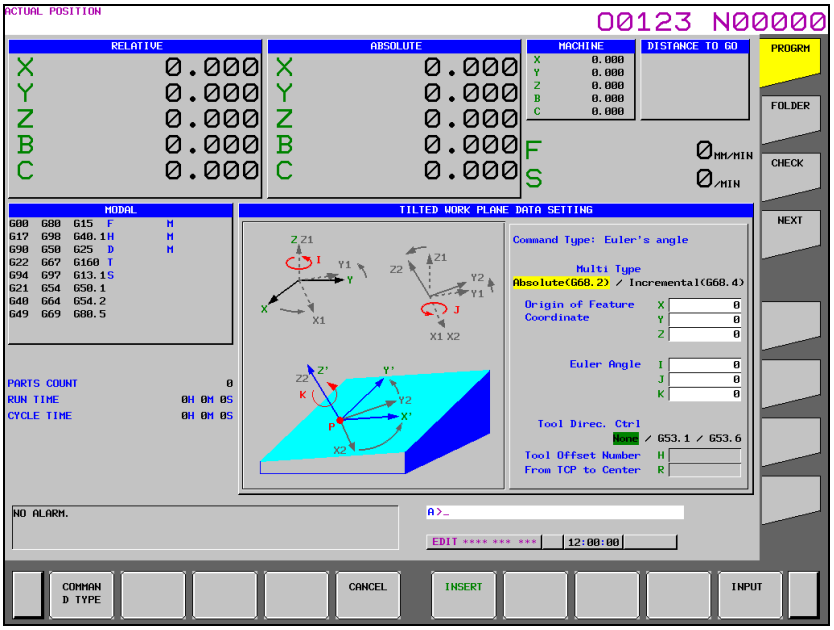


2 Press any of the cursor keys to move the cursor to a position where you want to insert a block.
Note that a block created on the guidance screens is inserted after the block at the cursor position. (If the block at the cursor position includes a tilted working plane indexing, the existing block is modified. See "Modification to an existing block" below.)

3 Press continuous menu key ▷ several times, and then press horizontal soft key [GUIDANCE TWP].
The command type selection screen is displayed.

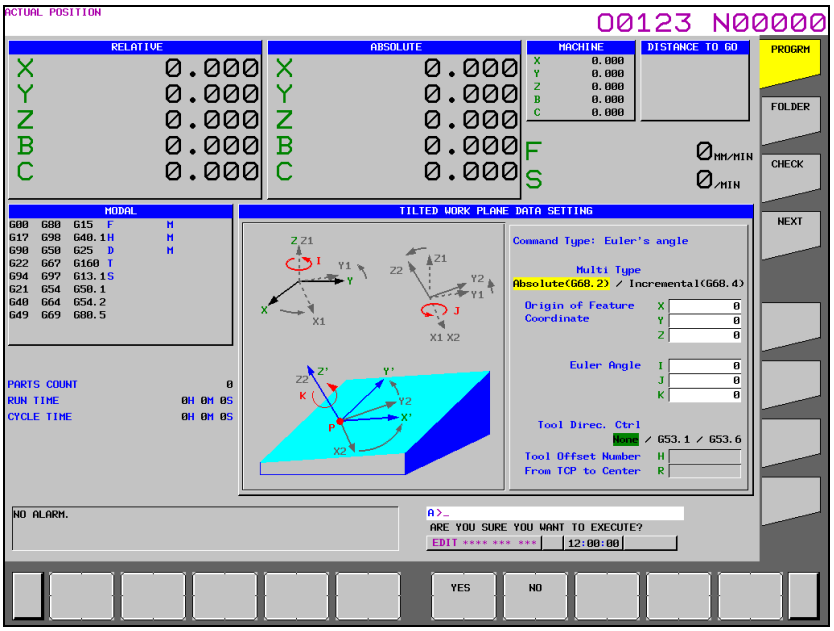


4 Select a command type with any of the cursor keys, and then press horizontal soft key [SELECT].
The tilted working plane data setting screen is displayed.



5 Enter command data for the setting items.

6 Press horizontal soft key [INSERT].

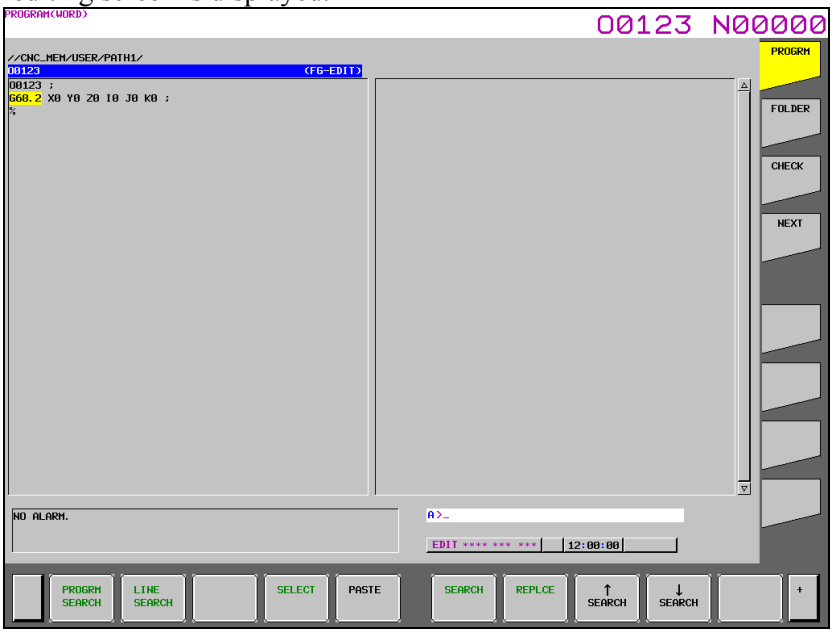


7 Press horizontal soft key [YES].
This takes you back to the program editing screen, where the new block is inserted after the block at the cursor position.

Modification to an existing block

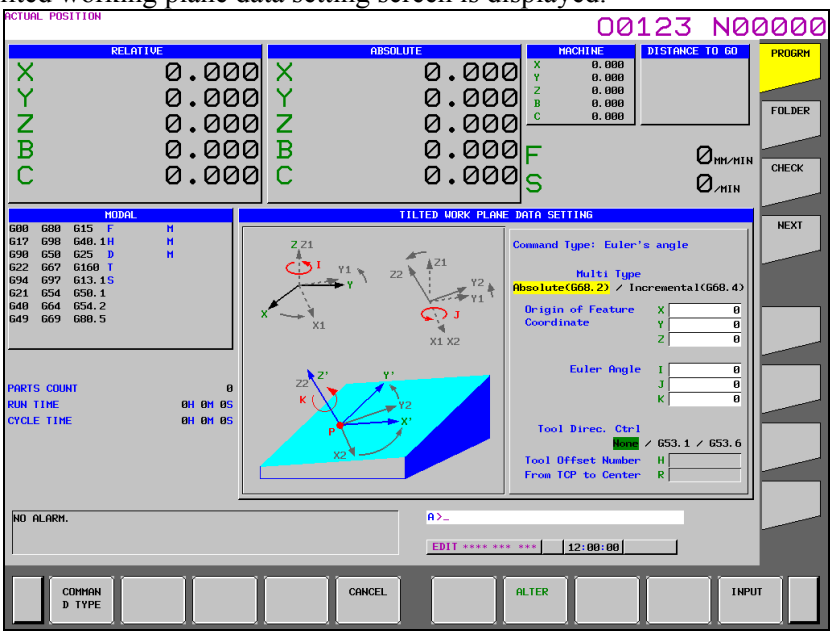
The following describes the procedure for replacing a block in a program being edited on a program editing screen, with a tilted working plane indexing block created on a guidance screen.

1 On a program editing screen, display a program to be edited.
(For the procedure for displaying a program editing screen, see step 1 in "Creation of a new block".)
The program editing screen is displayed.



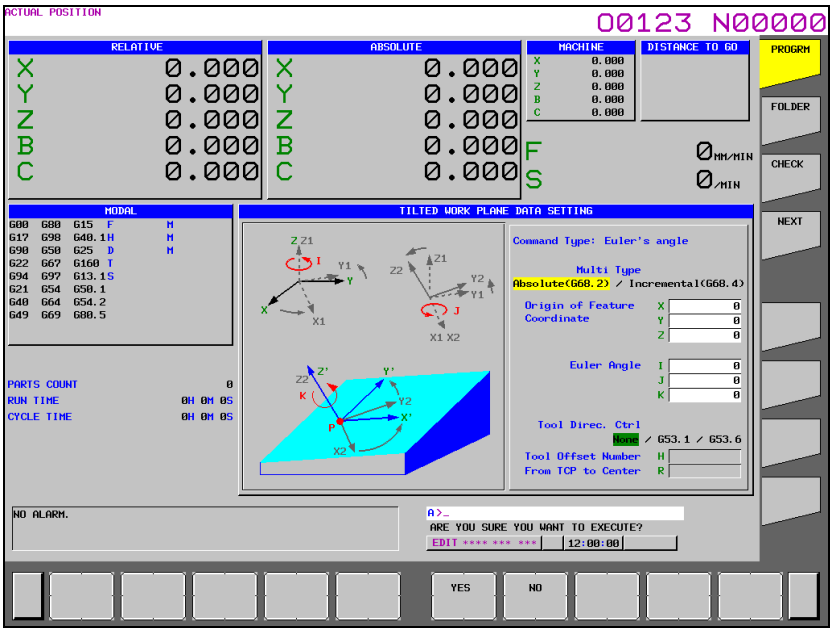
2 Press any of the cursor keys to move the cursor to a block to be modified.
For a command extending over more than one block, move the cursor to the first block.

3 Press continuous menu key ▷ several times, and then press horizontal soft key [GUIDANCE TWP]. The tilted working plane data setting screen is displayed.



4 Enter command data for setting items to be modified.

5 Press horizontal soft key [ALTER].



6 Press horizontal soft key [YES].
This takes you back to the program editing screen, where the block at the cursor position is replaced.

Guidance screen cancellation

Pressing horizontal soft key [CANCEL] on a guidance screen takes you back to the program editing screen. At this time, the data that has been set on the guidance screen is discarded.

NOTE

1 In addition to the above operation, the following operations also cancel a guidance screen. The data that has been set on the guidance screen is discarded.
- When bit 7 (CPG) of parameter No. 11302 is 1 (setting for automatically switching between program-related screens according to CNC mode), the CNC mode is changed.
- When a guidance screen is displayed from the foreground editing screen, the CNC mode is changed to a mode other than EDIT, TJOG, or THND.
- When a guidance screen is displayed from the MDI editing screen, the CNC mode is changed to a mode other than MDI.
- When a guidance screen is displayed on a 15/19-inch display unit, the screen is switched by a vertical soft key.
- The screen is switched by an MDI key.
- The screen is switched by a path select signal.
- An event that causes screen switching has occurred, including occurrence of an alarm, display of an operator message, or signal-based display of a screen (such as a tool compensation screen, workpiece shift screen, workpiece coordinate system setting screen, or C Language Executor screen).

2 If MDI key 

is pressed after switching from a guidance screen to another screen, a program editing screen is displayed, instead of the guidance screen.

Notes

- Conditions under which horizontal soft key [GUIDANCE TWP] is displayed
  Horizontal soft key [GUIDANCE TWP] is displayed on a program editing screen under the following conditions:
  1 Foreground editing screen
    - The CNC mode is EDIT, TJOG, or THND.
    - The editing and display are not prohibited for a program to be edited.
  2 Background editing screen
    - A program to be edited is not in reference mode.
    - The editing and display are not prohibited for a program to be edited.
  3 MDI editing screen
    - The CNC mode is MDI.

- Screen displayed when horizontal soft key [GUIDANCE TWP] is pressed
  When horizontal soft key [GUIDANCE TWP] is pressed, which guidance screen is displayed is automatically determined according to the conditions described below.
  1 When a tilted working plane indexing is not included in a block at the cursor position on a program editing screen
    The command type selection screen is displayed. A block created on the guidance screens is inserted after the block at the cursor position on the program editing screen.
  2 When a tilted working plane indexing is included in a block at the cursor position on a program editing screen
    The tilted working plane data setting screen is displayed, which shows the data for the tilted working plane indexing in the block at the cursor position on the program editing screen. When a block of a tool axis direction control command exists immediately after the tilted working plane indexing, the command data for the block is also shown. The tilted working plane indexing in the block at the cursor position on the program editing screen is replaced with a block created on the guidance screen.

NOTE
If the CNC is in the reset state or emergency stop state when horizontal soft key [GUIDANCE TWP] is pressed on the foreground editing screen or MDI editing screen, the warning "PROGRAM READ FAILED" appears, and the operation cannot be continued. (Horizontal soft keys other than [CANCEL] are not displayed.) Press horizontal soft key [CANCEL] to return to the program editing screen, and then press horizontal soft key [GUIDANCE TWP] again.

12.2.18.1 Command type selection screen

The command type selection screen is used to select the type of a tilted working plane indexing you want to insert into a program to be edited. One of the following command types can be selected:

- G68.2 / G68.4 (Euler's angle)
- G68.2 / G68.4 P1 (Roll-Pitch-Yaw angle)
- G68.2 / G68.4 P2 (3 points specification)
- G68.2 / G68.4 P3 (2 vectors specification)
- G68.2 / G68.4 P4 (Projection angle)
- G68.3 (Tool Axis Direction)

G68.2 is an absolute command, and G68.4 is an incremental command.

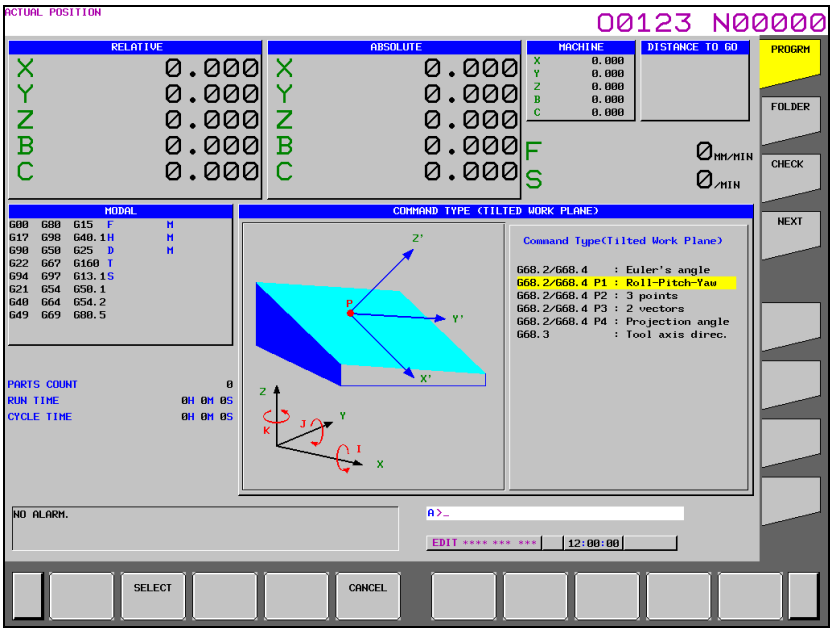


Fig. 12.2.18.1 (a) Command type selection screen (15-inch display unit)

Selection of a command type

1 Press cursor key ↑ or ↓ to move the cursor to a command type you want to select. As the cursor is moved, the figure corresponding to the command type at the cursor position is displayed.

2 When horizontal soft key [SELECT] is pressed, the command type at the cursor position is accepted, and the tilted working plane data setting screen is displayed.

NOTE
If the warning "PROGRAM READ FAILED" appears when the command type selection screen is displayed, the operation cannot be continued. (Horizontal soft keys other than [CANCEL] are not displayed.) Press horizontal soft key [CANCEL] to return to the program editing screen, and then press horizontal soft key [GUIDANCE TWP] again.

12.2.18.2 Tilted working plane data setting screen

The tilted working plane data setting screen is used to set specified tilted working plane data required for a tilted working plane indexing of the type that has been selected on the command type selection screen or has been selected when horizontal soft key [GUIDANCE TWP] is pressed.
Different types of tilted working plane data setting screen are provided for different command types. For details of each type of tilted working plane data setting screen, see "Details of the tilted working plane data setting screen".

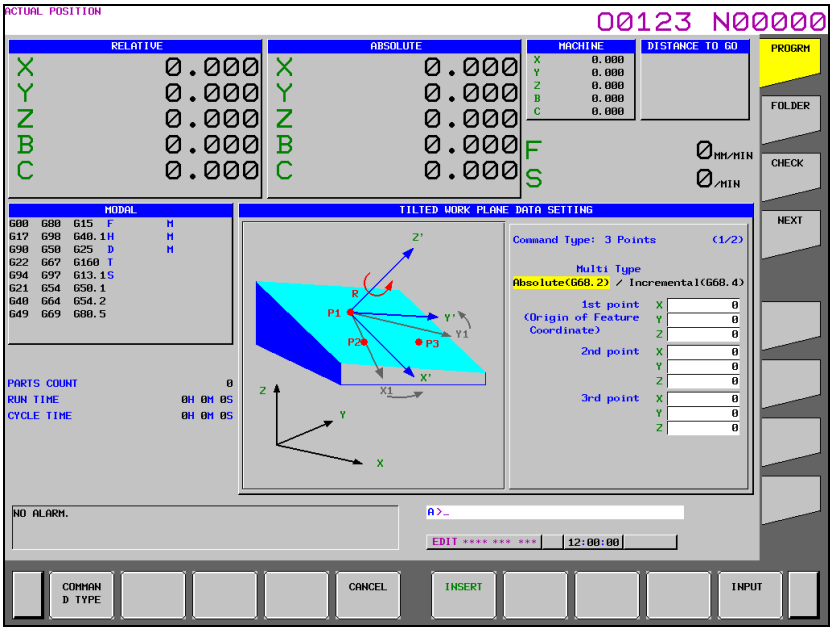


Fig. 12.2.18.2 (a) Tilted working plane data setting screen-3 points specification(15-inch display unit)

Display of the tilted working plane data setting screen

When a new block is created, the initial values are set in the setting and selection items.
When an existing block is modified, the command data for the block at the cursor position on the program editing screen is reflected in the setting and selection items.

NOTE
1 For existing block modification, if the guidance screen is displayed when the cursor is placed in the middle of multiple blocks for a command, the parameters for the block(s) before the cursor are not reflected in the setting and selection items, but only the parameters for the block(s) after the cursor are reflected. If no data is to be reflected in a setting or selection item, the initial value is set in the item.
2 For existing block modification, only the commands with parameter-specified axis names are reflected in the setting items on the screen. (The values specified in data for commands with incorrect axis names are not reflected in the setting items.)

Command data input

- Item for which to enter a value

  Press cursor key [↑] or [↓] to move the cursor to an item you want to set.

  Enter a value, and then press the [INPUT] key or horizontal soft key [INPUT].



  Example)

  When the origin of a feature coordinate system is set as shown above, addresses X, Y, and Z are specified as follows:

  G68.2 X0.001 Y0.01 Z1000 ...

  For two items "Tool Offset Number" and "From TCP to center", the state with no command can be set by deleting the specified values according to the following procedure.

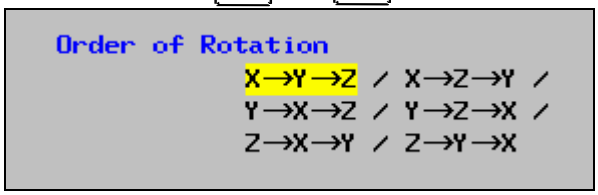
  1 Press cursor key [↑] or [↓] to move the cursor to the item "Tool Offset Number" or "From TCP to Center".

  2 Press [INPUT] or horizontal soft key [INPUT] without entering anything.

- Item to be selected from a list

  1 Press cursor key [↑] or [↓] to move the cursor to an item you want to set.

  2 Press cursor key [←] or [→] to move the cursor to an item you want to select.



  Example)

  Order of rotation for Roll-Pitch Yaw angle

Insertion of a block

If the guidance screen is displayed when the block at the cursor position on the program editing screen does not include a tilted working plane indexing, horizontal soft key [INSERT] is displayed on the tilted working plane data setting screen. An operation for inserting a tilted working plane indexing block is described below.

For warnings that may be issued at the time of block insertion, see "Limitation".

1 Press horizontal soft key [INSERT].
The confirmation message "ARE YOU SURE YOU WANT TO EXECUTE?" is displayed, and horizontal soft keys [YES] and [NO] are displayed.

2 Press horizontal soft key [YES].
A block is created from the command type and command data and then is inserted after the block at the cursor position in the program to be edited. After block insertion, the guidance screen is closed and the program editing screen is displayed.
Pressing horizontal soft key [NO] takes you back to the tilted working plane data setting screen.

Example)

G00 X0.;

↓

G00 X0.;
G68.2 P2 Q0...
G68.2 P2 Q1...
G68.2 P2 Q2...
G68.2 P2 Q3...

When the guidance screen is displayed and the 3-point specification is selected as a command type for block insertion, a created block is inserted after the block at the cursor position.

Replacement of a block

If the guidance screen is displayed when the block at the cursor position on the program editing screen includes a tilted working plane indexing, horizontal soft key [ALTER] is displayed on the tilted working plane data setting screen. An operation for replacing a tilted working plane indexing block is described below.
For warnings that may be issued at the time of block replacement, see "Limitation".

1 Press horizontal soft key [ALTER].
The confirmation message "ARE YOU SURE YOU WANT TO EXECUTE?" is displayed, and horizontal soft keys [YES] and [NO] are displayed.
2 Press horizontal soft key [YES].
A block is created from the command type and command data and then replaces the tilted working plane indexing in the block at the cursor position on the program editing screen. After block replacement, the guidance screen is closed and the program editing screen is displayed.
Pressing horizontal soft key [NO] takes you back to the tilted working plane data setting screen.

NOTE
1 If a block to be replaced includes a command other than a tilted working plane indexing, the command is deleted during block replacement. However, only the sequence number at the beginning is preserved.
2 If the guidance screen is displayed when the cursor is placed in the middle of multiple blocks for a command, the block(s) before the cursor are not replaced. These blocks remain unchanged after replacement.
3 If horizontal soft key [GUIDANCE TWP] is pressed when the cursor is placed on a block of a tool axis direction control command, the command type selection screen is displayed in new insertion mode. A block created on the guidance screens is inserted after the block of the tool axis direction control command.

Limitation

The following lists the warnings that may be issued at the time of block insertion or replacement.
If a warning is displayed, return to the program editing screen with horizontal soft key [CANCEL] and press horizontal soft key [GUIDANCE TWP] again, or eliminate the cause of the warning and retry the operation.

Warning	Description
“PROGRAM WRITE FAILED”	• The guidance screen was displayed from the foreground editing screen or MDI editing screen, and a block insertion or replacement operation was performed when the CNC was in the reset state or emergency stop state.
“PROGRAM CANNOT BE WRITTEN”	• A block insertion or replacement operation was performed after a program to be edited was updated due to program downloading by an external application while the guidance screen was displayed. • A block insertion or replacement operation was performed after the start of the main program was located by a reset while the guidance screen was displayed.

Warning	Description
"WRITE PROTECT"	• A block insertion or replacement operation was performed when the editing or display was prohibited for a program to be edited. • A block insertion or replacement operation was performed when the memory protection signal (KEY signal) for program registration or editing was off. • A block insertion or replacement operation was performed when the operation level of the 8-level data protection function was lower than the protection level of a part program editing operation.

Display of the command type selection screen

When horizontal soft key [COMMAND TYPE] is pressed, the command type selection screen is displayed. When the command type is changed on the command type selection screen, the values set on the tilted working plane data setting screen are cleared.

When a tilted working plane indexing is included in the block at the cursor position on the program editing screen, you can display the guidance screen, change the command type on the command type selection screen, and then perform block replacement. In this case, the block at the cursor position on the program editing screen is replaced with a block created from the command type and command data that have been set.

12.2.18.3 Details of the tilted working plane data setting screen

The following six tilted working plane indexing are supported.
For details of the commands, see II-22.3, "TILTED WORKING PLANE INDEXING".

- G68.2 / G68.4 (Euler's angle)
- G68.2 / G68.4 P1 (Roll-Pitch-Yaw angle)
- G68.2 / G68.4 P2 (3 points specification)
- G68.2 / G68.4 P3 (2 vectors specification)
- G68.2 / G68.4 P4 (Projection angle)
- G68.3 (Tool Axis Direction)

G68.2 is an absolute command, and G68.4 is an incremental command.

G68.2 / G68.4(Euler's angle)

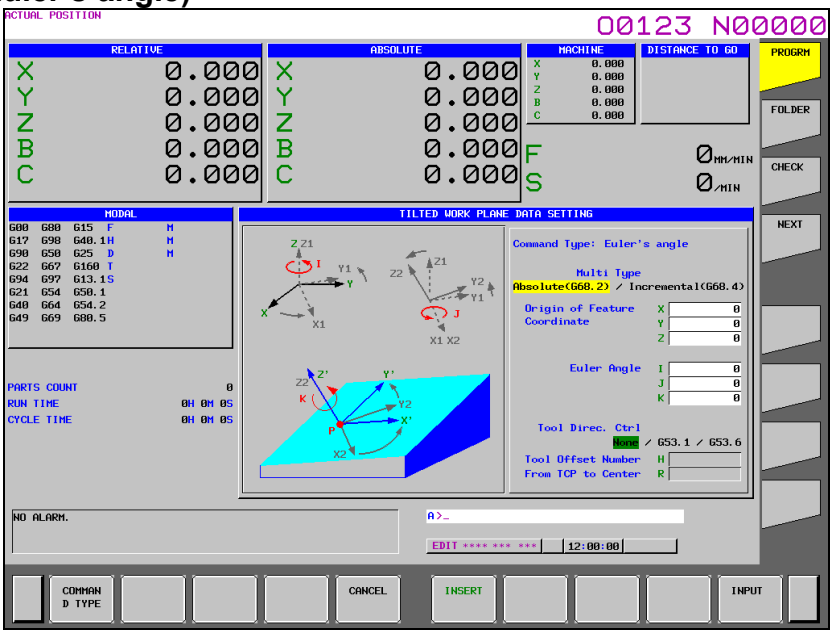


Fig. 12.2.18.3 (a) Tilted working plane data setting screen-Euler's angle(15-inch display unit)

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Euler's angle
  I : Specify an angle of rotation around the Z-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).
  This rotation determines coordinate system 1 (X1-Y1-Z1) from the workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type) (X-Y-Z).
  J : Specify an angle of rotation around the X-axis of coordinate system 1.
  This rotation determines coordinate system 2 (X2-Y2-Z2) from coordinate system 1 (X1-Y1-Z1).
  K : Specify an angle of rotation around the Z-axis of coordinate system 2.
  After this rotation, a feature coordinate system is obtained by shifting the origin of the workpiece coordinate system by the coordinates specified in "Origin of Feature Coordinate".

G68.2 / G68.4(Roll-Pitch-Yaw angle)

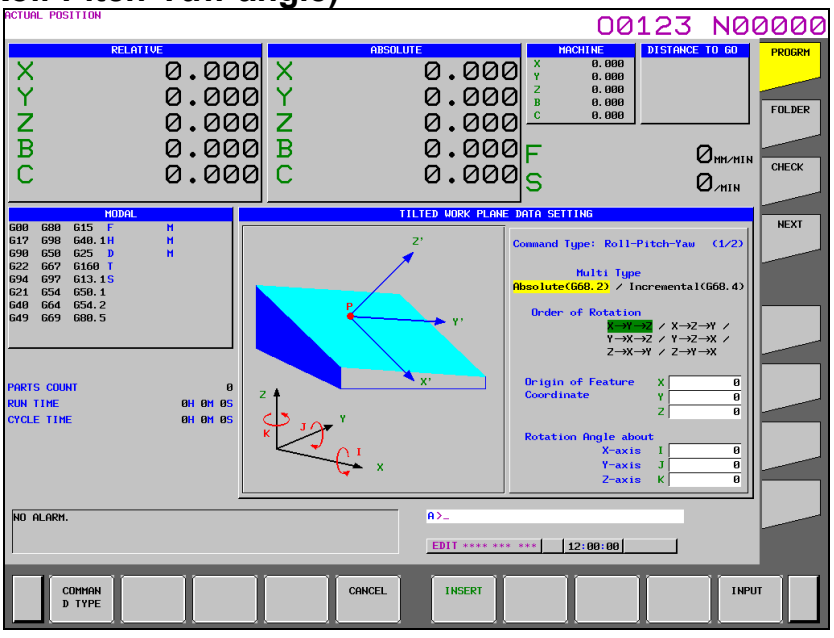


Fig. 12.2.18.3 (b) Tilted working plane data setting screen- Roll-Pitch-Yaw angle(15-inch display unit)

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Order of Rotation
  Select an order in which the X-axis, Y-axis, and Z-axis are rotated in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). The selectable rotation orders are as follows:

	1st rotation axis	2nd rotation axis	3rd rotation axis
X→Y→Z	X-axis	Y-axis	Z-axis
X→Z→Y	X-axis	Z-axis	Y-axis
Y→X→Z	Y-axis	X-axis	Z-axis
Y→Z→X	Y-axis	Z-axis	X-axis
Z→X→Y	Z-axis	X-axis	Y-axis
Z→Y→X	Z-axis	Y-axis	X-axis

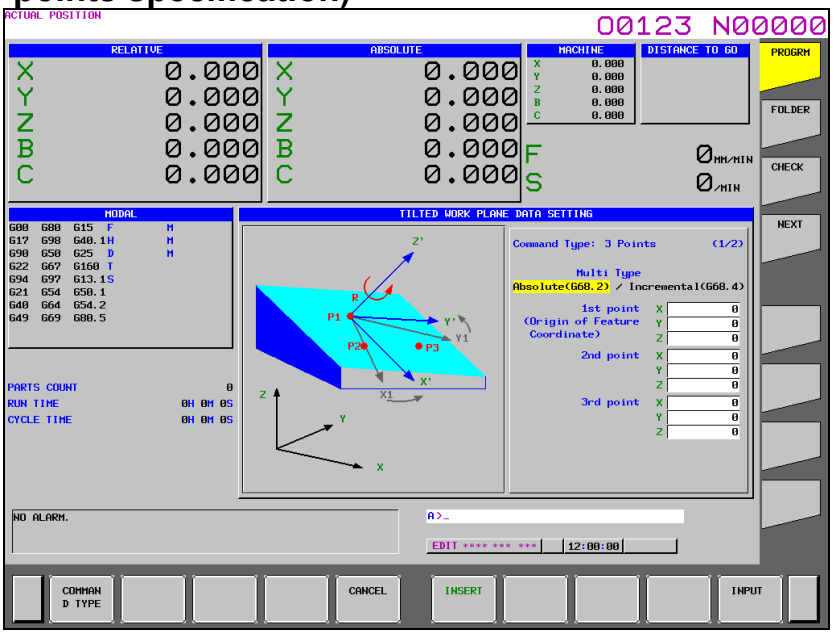
- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about X-axis
  Specify an angle of rotation around the X-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about Y-axis
  Specify an angle of rotation around the Y-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Rotation Angle about Z-axis
  Specify an angle of rotation around the Z-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

G68.2 / G68.4(3 points specification)



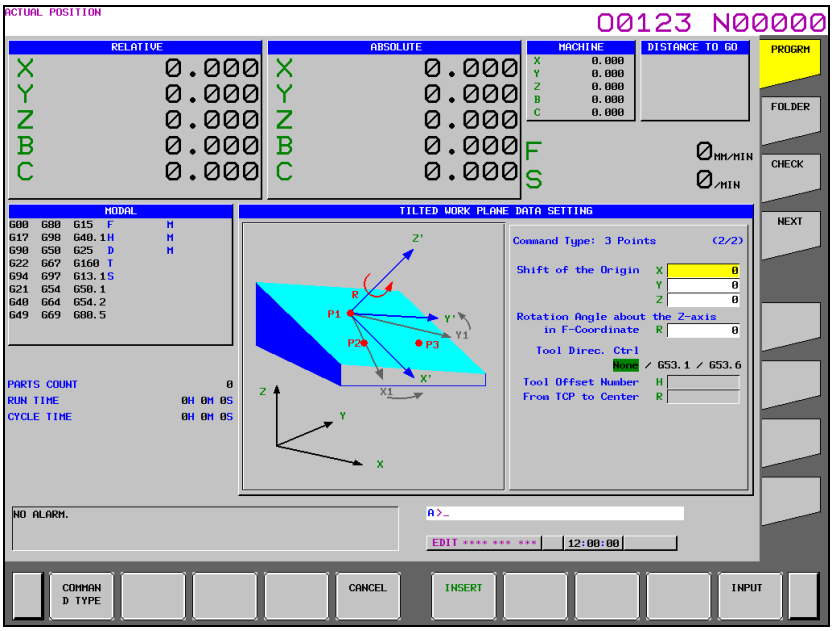


Fig. 12.2.18.3 (c) Tilted working plane data setting screen- 3 points specification(15-inch display unit)

- Multi Type
  Absolute:
  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.
  Incremental:
  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Shift of the Origin
  Specify, in a feature coordinate system, an amount of shift from the feature coordinate system origin specified for the 1st point (point P1).

- Rotation Angle about the Z-axis in F-Coordinate
  Specify an angle of rotation around the Z-axis of a feature coordinate system. The direction of rotation angle R is positive when a rotation is made clockwise as viewed in the Z-axis direction of the feature coordinate system.

- 1st point (Origin of Feature Coordinate)
  Specify the origin (X, Y, and Z of point P1) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- 2nd point
  Specify the coordinates (X, Y, and Z of point P2) of the 2nd point as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). The 1st point and 2nd point determine the X-axis of the feature coordinate system.

- 3rd point
  Specify the coordinates (X, Y, and Z of point P3) of the 3rd point as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type). Among the directions normal to the X-axis, a direction with a smaller angle relative to the P1 → P3 vector is the Y-axis of the feature coordinate system.

G68.2 / G68.4(2 vectors specification)

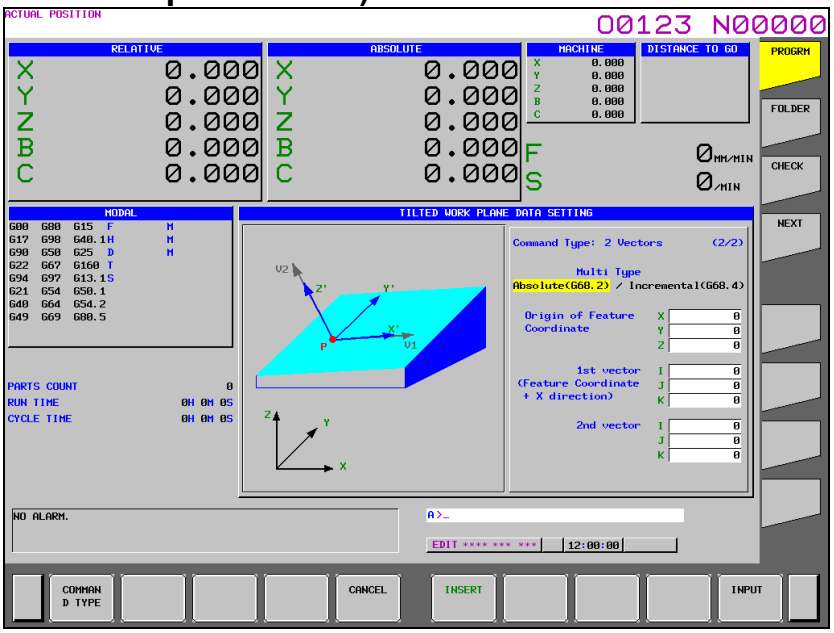


Fig. 12.2.18.3 (d) Tilted working plane data setting screen-2 vectors specification(15-inch display unit)

- Multi Type

  Absolute:

  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.

  Incremental:

  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate

  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- V1 (X') vector

  Specify the X-axis direction vector of a feature coordinate system as values in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- V2 (Z') vector

  Specify the Z-axis direction vector of a feature coordinate system as values in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

G68.2 / G68.4(Projection angle)

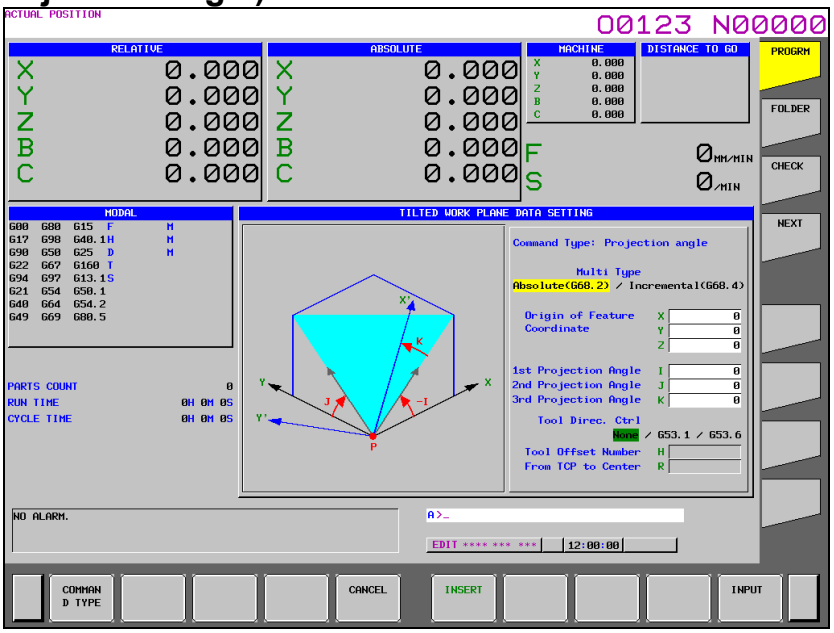


Fig. 12.2.18.3 (e) Tilted working plane data setting screen-Projection angle(15-inch display unit)

- Multi Type

  Absolute:

  It is assumed that values of specified data are in a workpiece coordinate system, regardless of whether tilted working plane indexing mode is set.

  Incremental:

  It is assumed that values of specified data are in a feature coordinate system if tilted working plane indexing mode is already set.

- Origin of Feature Coordinate

  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

- Projection angle

  I : Specify a projection angle relative to the X-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

  J : Specify a projection angle relative to the Y-axis of a workpiece coordinate system (for the absolute type) or the current feature coordinate system (for the incremental type).

  K : Specify an angle of rotation around the Z-axis of a feature coordinate system.

G68.3(Tool Axis Direction)

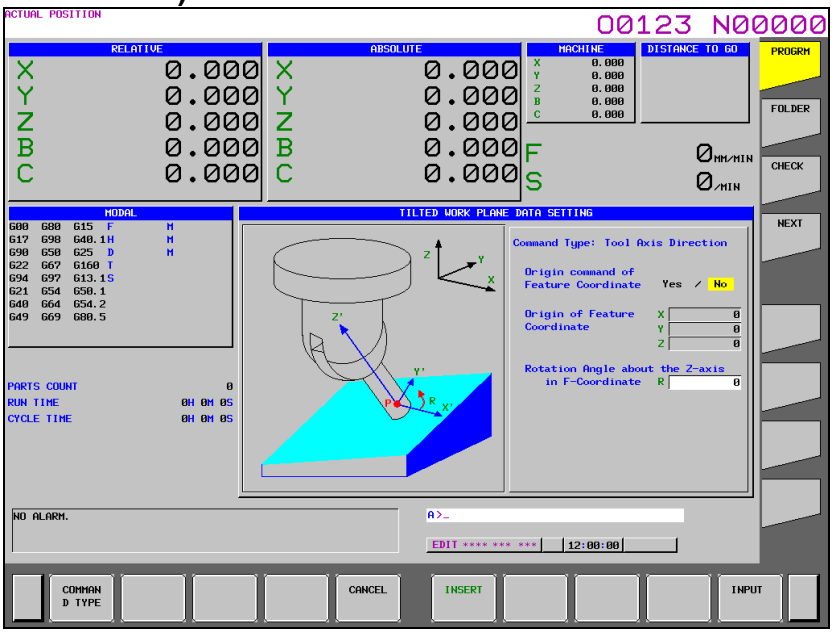


Fig. 12.2.18.3 (f) Tilted working plane data setting screen-Tool Axis Direction(15-inch display unit)
(When "No" is selected in "Origin command of Feature Coordinate")

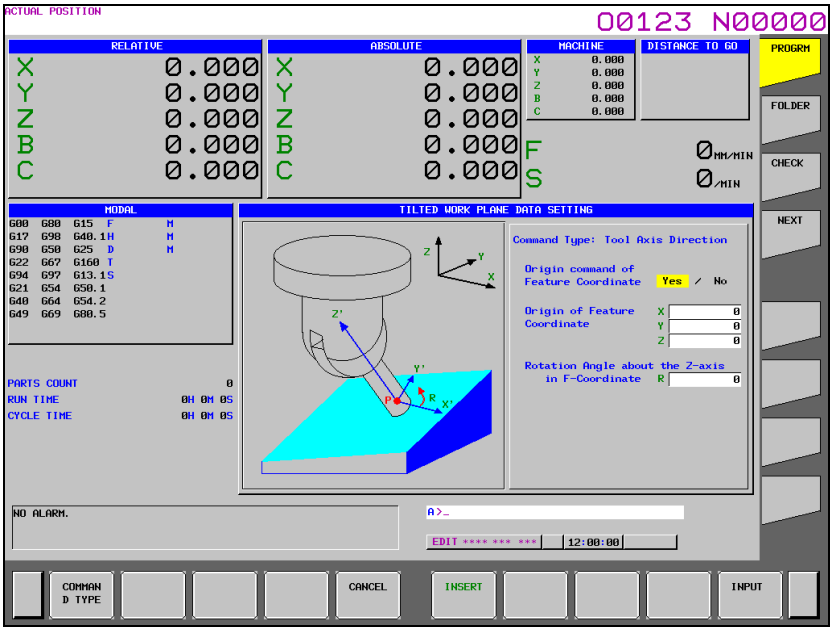


Fig. 12.2.18.3 (g) Tilted working plane data setting screen-Tool Axis Direction(15-inch display unit)
(When "Yes" is selected in "Origin command of Feature Coordinate")

- Origin command of Feature Coordinate
  Select whether to specify a feature coordinate system origin.
  Yes : A feature coordinate system origin is specified.
  No : A feature coordinate system origin is not specified.

- Origin of Feature Coordinate
  Specify the origin (X, Y, and Z of point P) of a feature coordinate system as coordinates in a workpiece coordinate system.
  This setting cannot be made if "No" is selected in "Origin command of Feature Coordinate".

- Rotation Angle about the Z-axis in F-Coordinate
  Specify an angle of rotation around the Z-axis of a feature coordinate system. The direction of rotation angle R is positive when a rotation is made clockwise as viewed in the Z-axis direction of the feature coordinate system.

12.2.18.4 Limitation

This function is supported on 10.4-inch, 15-inch, and 19-inch display units. This function is not displayed on 8.4-inch display unit.

12.3 SCREENS DISPLAYED BY FUNCTION KEY

Section 12.3, "SCREENS DISPLAYED BY FUNCTION KEY", consists of the following subsections:

——————— Screens of a 8.4/10.4-inch display unit

12.3.1 Displaying and Entering Setting Data....1458
12.3.2 Sequence Number Comparison and Stop....1461
12.3.3 Displaying and Setting Run Time, Parts Count, and Time....1462
12.3.4 Displaying and Setting the Workpiece Origin Offset Value....1464
12.3.5 Direct Input of Workpiece Origin Offset Value Measured....1465
12.3.6 Displaying and Setting Custom Macro Common Variables....1466
12.3.7 Displaying and Setting Real Time Custom Macro Data....1468
12.3.8 Displaying and Setting the Software Operator's Panel....1469
12.3.9 Setting and Displaying Tool Management Data....1472
12.3.10 Displaying and Switching the Display Language....1488
12.3.11 Protection of Data at Eight Levels....1490
12.3.12 Precision Level Selection....1496
12.3.13 Machining Level Selection....1498
12.3.14 Machining Quality Level Selection....1499
12.3.15 Displaying and Setting Tool Life Management Data....1500
12.3.16 Displaying and Setting Workpiece Setting Error Compensation Data....1511
12.3.17 Displaying and Setting Pattern Data Inputs....1512
12.3.18 Setting and Displaying Data When the Tool Offset for Milling and Turning Function Is Enabled....1514

——————— Screens of a 15/19-inch display unit

12.3.19 Displaying and Entering Setting Data (15/19-inch Display Unit)....1517
12.3.20 Sequence Number Comparison and Stop (15/19-inch Display Unit)....1519
12.3.21 Displaying and Setting Run Time, Parts Count, and Time (15/19-inch Display Unit)....1521
12.3.22 Displaying and Setting the Workpiece Origin Offset Value (15/19-inch Display Unit)....1523
12.3.23 Direct Input of Workpiece Origin Offset Value Measured (15/19-inch Display Unit)....1524
12.3.24 Displaying and Setting Custom Macro Common Variables (15/19-inch Display Unit)....1525
12.3.25 Displaying and Setting Real Time Custom Macro Data (15/19-inch Display Unit)....1527
12.3.26 Displaying and Setting the Software Operator's Panel (15/19-inch Display Unit)....1528
12.3.27 Setting and Displaying Tool Management Data (15/19-inch Display Unit)....1531
12.3.28 Displaying and Switching the Display Language (15/19-inch Display Unit)....1549
12.3.29 Protection of Data at Eight Levels (15/19-inch Display Unit)....1550
12.3.30 Precision Level Selection (15/19-inch Display Unit)....1556
12.3.31 Machining Level Selection (15/19-inch Display Unit)....1558
12.3.32 Machining Quality Level Selection (15/19-inch Display Unit)....1559
12.3.33 Displaying and Setting Tool Life Management Data (15/19-inch Display Unit)....1561
12.3.34 Displaying and Setting Workpiece Setting Error Compensation Data (15/19-inch Display Unit)....1572
12.3.35 Displaying and Setting Pattern Data Inputs (15/19-inch Display Unit)....1573
12.3.36 Built-in 3D Interference Check....1576
12.3.37 Setting and Displaying Data When the Tool Offset for Milling and Turning Function Is Enabled (15/19-inch Display Unit)....1624

Screens of a 8.4/10.4-inch display unit

Press function key  to display or set tool compensation values and other data.

This section describes how to display or set the following data:

1. Tool compensation value
2. Settings
3. Sequence number comparison and stop
4. Run time and part count
5. Workpiece origin offset value
6. Custom macro common variables
7. Software operator's panel
8. Tool management data
9. Display language switching
10. Protection of data at eight levels
11. Precision level selection
12. Tool life management data
13. Workpiece setting error compensation data
14. Pattern data input

The software operator's panel, display language switching, precision level selection, and pattern data input depend on the specifications of the machine tool builder. See the manual issued by the machine tool builder for details.

12.3.1 Displaying and Entering Setting Data

Data such as the TV check flag and punch code is set on the setting data screen. On this screen, the operator can also enable/disable parameter writing, enable/disable the automatic insertion of sequence numbers in program editing, and perform settings for the sequence number comparison and stop function.
See III-9.2 for automatic insertion of sequence numbers.
See III-12.3.2 for the sequence number comparison and stop function. This subsection describes how to set data.

Procedure for setting the setting data

Procedure

1 Select the MDI mode.

2 Press function key .

3 Press soft key [SETTING] to display the setting data screen.
This screen consists of several pages.
Press page key  or  until the desired screen is displayed.
An example of the setting data screen is shown below.

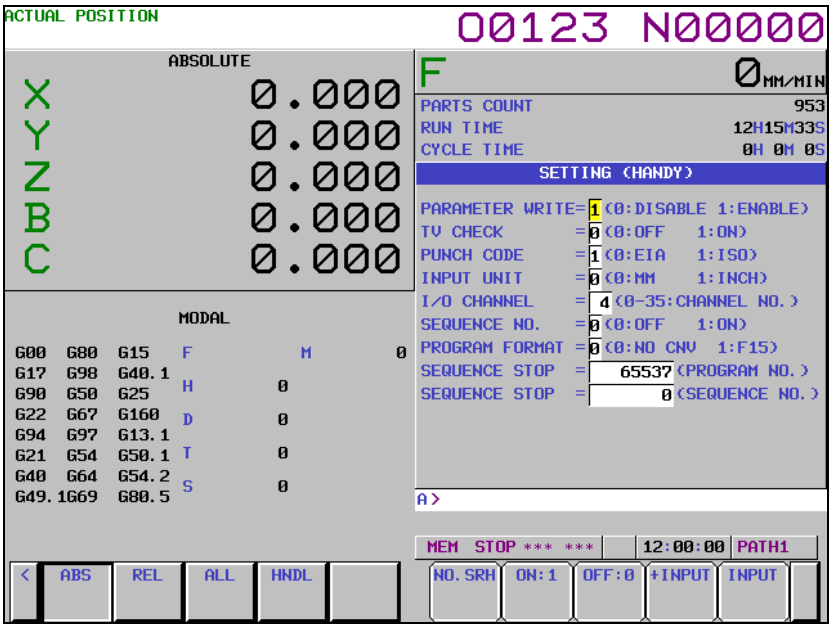


Fig. 12.3.1 (a) SETTING (HANDY) screen (10.4-inch display unit)

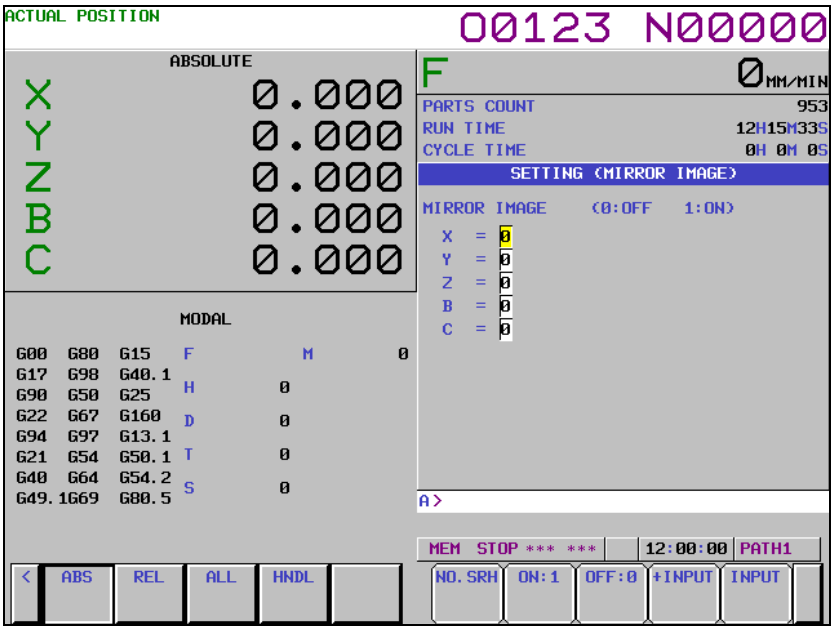
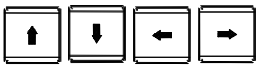


Fig. 12.3.1 (b) SETTING (MIRROR IMAGE) screen (10.4-inch display unit)

4 Move the cursor to the item to be changed by pressing cursor keys .

5 Enter a new value and press soft key [INPUT].

Explanation

- PARAMETER WRITE

Setting whether parameter writing is enabled or disabled.

0 : Disabled

1 : Enabled

- TV CHECK

Setting to perform TV check.
0 : No TV check
1 : Perform TV check

- OUTPUT CODE

Setting code when data is output through RS232C interface.
0 : EIA code output
1 : ISO code output

- INPUT UNIT

Setting a program input unit, inch or metric system
0 : Metric
1 : Inch

- I/O CHANNEL

Using channel of RS232C interface.
0 : Channel 0
1 : Channel 1
2 : Channel 2

- SEQUENCE NO.

Setting of whether to perform automatic insertion of the sequence number or not at program edit in the EDIT mode.
0 : Does not perform automatic sequence number insertion.
1 : Perform automatic sequence number insertion.

- PROGRAM FORMAT

Setting of whether to use the Series 15 format.
0: Uses the standard format.
1: Uses the Series 15 format.
For the Series 15 format, refer to Chapter II-6 in the OPERATOR'S MANUAL (T series) or Chapter II-7 of Part II in the OPERATOR'S MANUAL (M series).

- SEQUENCE STOP

Setting the sequence number with which the operation stops for the sequence number comparison and stop function and the number of the program to which the sequence number belongs

- MIRROR IMAGE

Setting of mirror image ON/OFF for each axes.
0 : Mirror image off
1 : Mirror image on

- Others

Page key 

or 

can also be pressed to display the SETTING (TIMER) screen. See III-12.3.3 for this screen.

12.3.2 Sequence Number Comparison and Stop

If a block containing a specified sequence number appears in the program being executed, operation enters single block mode after the block is executed.

Procedure for sequence number comparison and stop

Procedure

1 Select the MDI mode.

2 Press function key .

3 Press chapter selection soft key [SETTING].

4 Press page key  or  several times until the following screen is displayed.

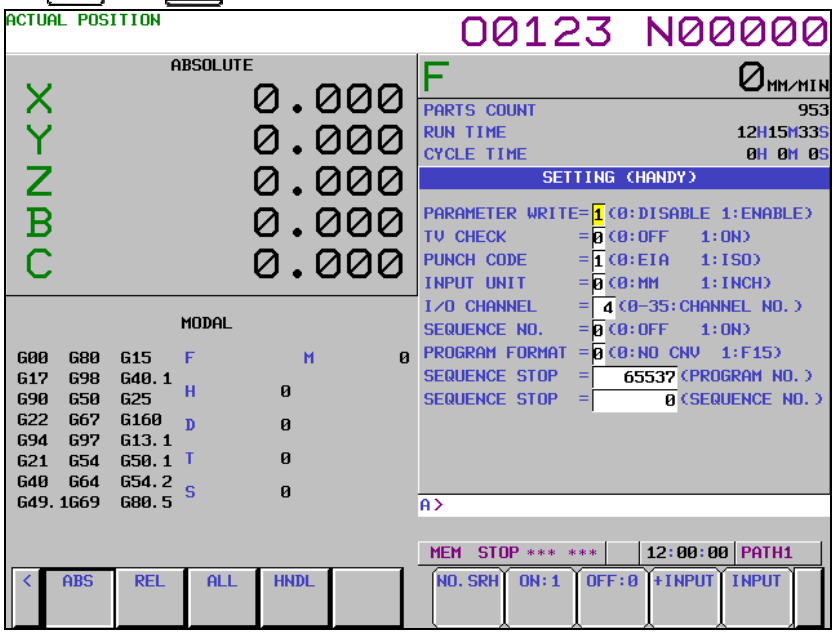


Fig. 12.3.2 (a) SETTING (HANDY) screen (10.4-inch display unit)

5 Enter in (PROGRAM NO.) for SEQUENCE STOP the number (1 to 99999999) of the program containing the sequence number with which operation stops.

6 Enter in (SEQUENCE NO.) for SEQUENCE STOP (with eight or less digits) the sequence number with which operation is stopped.

7 When automatic operation is executed, operation enters single block mode at the block containing the sequence number which has been set.

Explanation

- Sequence number after the program is executed

After the specified sequence number is found during the execution of the program, the sequence number setting for sequence number compensation and stop becomes “-1”.

- Exceptional blocks

If the predetermined sequence number is found in a block in which all commands are those to be processed within the CNC control unit, the execution does not stop at that block.

```
[Example]  N1 #1=1 ;
           N2 IF[#1 EQ 1]GOTO 08 ;
           N3 GOTO 09 ;
           N4 M98 P1000 ;
           N5 M99 ;
```

In the example shown above, if the predetermined sequence number is found, the execution of the program does not stop.

- **Stop in the canned cycle**

If the predetermined sequence number is found in a block which has a canned cycle command, the execution of the program stops after the return operation is completed.

- **When the same sequence number is found several times in the program**

If the predetermined sequence number appears twice or more in a program, the execution of the program stops after the block in which the predetermined sequence number is found for the first time is executed.

- **Block to be repeated a specified number of times**

If the predetermined sequence number is found in a block which is to be executed repeatedly, the execution of the program stops after the block is executed specified times.

12.3.3 Displaying and Setting Run Time, Parts Count, and Time

Various run times, the total number of machined parts, number of parts required, and number of machined parts can be displayed.
This data can be set by parameters or on this screen (except for the total number of machined parts and the time during which the power is on, which can be set only by parameters).
This screen can also display the clock time. The time can be set on the screen.

Procedure for Displaying and Setting Run Time, Parts Count and Time

Procedure

1 Select the MDI mode.
2 Press function key .
3 Press chapter selection soft key [SETTING].
4 Press page key  or  several times until the following screen is displayed.

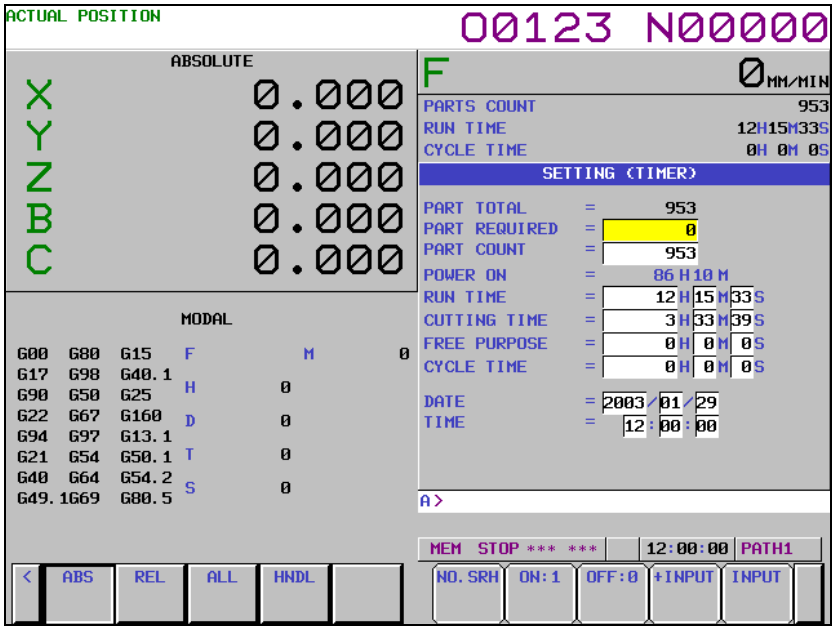


Fig. 12.3.3 (a) SETTING (TIMER) screen (10.4-inch display unit)

5 To set the number of parts required, move the cursor to PARTS REQUIRED and enter the number of parts to be machined.
6 To set the clock, move the cursor to DATE or TIME, enter a new date or time, then press soft key [INPUT].

Explanation

- PARTS TOTAL

This value is incremented by one when M02, M30, or an M code specified by parameter No. 6710 is executed. This value cannot be set on this screen. Set the value in parameter No. 6712.

- PARTS REQUIRED

It is used for setting the number of machined parts required.
When the “0” is set to it, there is no limitation to the number of parts.
Also, its setting can be made by the parameter No 6713.

- PARTS COUNT

This value is incremented by one when M02, M30, or an M code specified by parameter No. 6710 is executed. The value can also be set by parameter No. 6711. In general, this value is reset when it reaches the number of parts required. Refer to the manual issued by the machine tool builder for details.

- POWER ON

Displays the total time which the power is on. This value cannot be set on this screen but can be preset in parameter No. 6750.

- RUN TIME

Indicates the total run time during automatic operation, excluding the stop and feed hold time.
This value can be preset in parameter No. 6751 or No. 6752.

- CUTTING TIME

Displays the total time taken by cutting that involves cutting feed such as linear interpolation (G01) and circular interpolation (G02 or G03). This value can be preset in parameter No. 6753 or No. 6754.

- FREE PURPOSE

This value can be used, for example, as the total time during which coolant flows. Refer to the manual issued by the machine tool builder for details.

- CYCLE TIME

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- DATA and TIME

Displays the current date and time. The date and time can be set on this screen.

- Usage

When the command of M02 or M30 is executed, the total number of machined parts and the number of machined parts are incremented by one. Therefore, create the program so that M02 or M30 is executed every time the processing of one part is completed. Furthermore, if an M code set to the parameter No. 6710 is executed, counting is made in the similar manner. Also, it is possible to disable counting even if M02 or M30 is executed (bit 0 (PCM) of parameter No. 6700 is set to 1). For details, see the manual issued by machine tool builders.

Limitation

- Run time and part count settings

Negative value cannot be set. Also, the setting of "M" and "S" of run time is valid from 0 to 59.
Negative value may not be set to the total number of machined parts.

- Time settings

Neither negative value nor the value exceeding the value in the following table can be set.

Table 12.3.3 (a)

Item	Maximum value	Item	Maximum value
Year	2096	Hour	23
Month	12	Minute	59
Day	31	Second	59

12.3.4 Displaying and Setting the Workpiece Origin Offset Value

Displays the workpiece origin offset for each workpiece coordinate system (G54 to G59, G54.1 P1 to G54.1 P48 and G54.1 P1 to G54.1 P300) and external workpiece origin offset.
The workpiece origin offset value and external workpiece origin offset value can be set on this screen.

Procedure for displaying and setting the workpiece origin offset value

Procedure

1 Press function key .

2 Press chapter selection soft key [WORK].
The workpiece coordinate system setting screen is displayed.

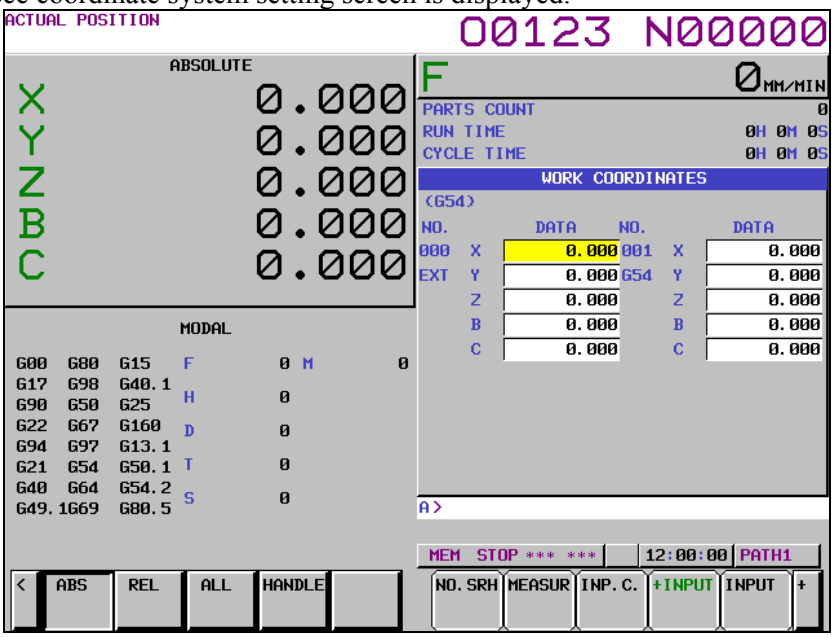


Fig. 12.3.4 (a) WORK COORDINATES screen (10.4-inch display unit)

3 The screen for displaying the workpiece origin offset values consists of two or more pages.
Display a desired page in either of the following two ways:

- Press the page key  or .
- Enter the workpiece coordinate system number (0 : external workpiece origin offset, 1 to 6: workpiece coordinate systems G54 to G59, P1 to P48 : workpiece coordinate systems G54.1 P1 to G54.1 P48, P1 to P300 : workpiece coordinate systems G54.1 P1 to G54.1 P300) and press operation selection soft key [NO.SRH].

4 Turn off the data protection key to enable writing.
5 Move the cursor to the workpiece origin offset to be changed.
6 Enter a desired value by pressing numeric keys, then press soft key [INPUT]. The entered value is specified in the workpiece origin offset value. Or, by entering a desired value with numeric keys and pressing soft key [+INPUT], the entered value can be added to the previous offset value.
7 Repeat steps 5 and 6 to change other offset values.
8 Turn on the data protection key to disable writing.

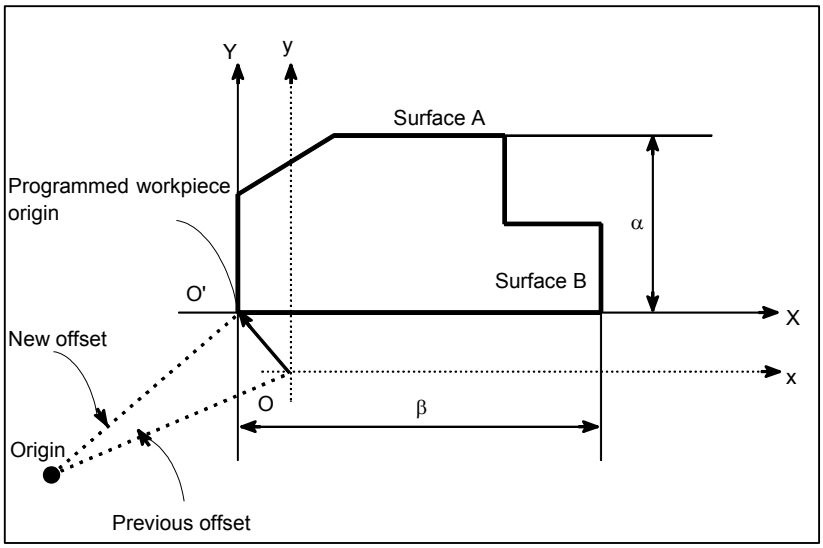
NOTE
When the axis name set in parameter No.1020 is not valid, the workpiece origin offset value of the axis should be set to 0.

12.3.5 Direct Input of Workpiece Origin Offset Value Measured

This function is used to compensate for the difference between the programmed workpiece coordinate system and the actual workpiece coordinate system. The measured offset for the origin of the workpiece coordinate system can be input on the screen such that the command values match the actual dimensions. Selecting the new coordinate system matches the programmed coordinate system with the actual coordinate system.

Procedure for direct input of workpiece origin offset value measured

Procedure



1 When the workpiece is shaped as shown above, position the reference tool manually until it touches surface A of the workpiece.
2 Retract the tool without changing the Y coordinate.
3 Measure distance α between surface A and the programmed origin of the workpiece coordinate system as shown above.
4 Press function key .
5 To display the WORK COORDINATES screen, press the chapter selection soft key [WORK].

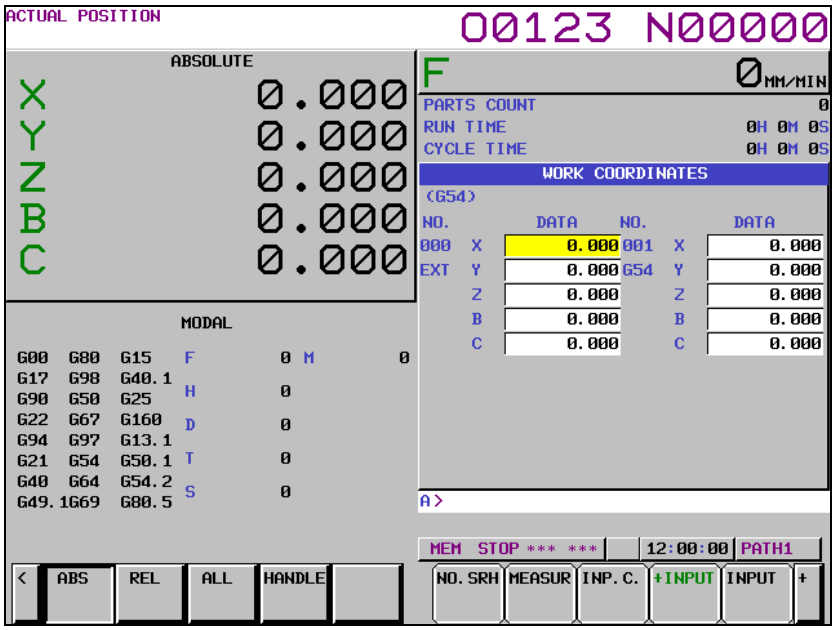


Fig. 12.3.5 (a) WORK COORDINATES screen (10.4-inch display unit)

6 Position the cursor to the workpiece origin offset value to be set.
7 Press the address key for the axis along which the offset is to be set (Y-axis in this example).
8 Enter the measured value (α) then press soft key [MEASUR].
9 Move the reference tool manually until it touches surface B of the workpiece.
10 Retract the tool without changing the X coordinate.
11 Enter the measured value of distance (β) to surface B as an X value as with steps 7 and 8.

Limitation

- Consecutive input

Offsets for two or more axes cannot be input at the same time.

- During program execution

This function cannot be used while a program is being executed.

12.3.6 Displaying and Setting Custom Macro Common Variables

Displays common variables (#100 to #149 or #100 to #199, and #500 to #531 or #500 to #999) on the screen.
The values for variables can be set on this screen.
Relative coordinates can also be set to variables.

Procedure for displaying and setting custom macro common variables

Procedure

1 Press function key .

2 Press the continuous menu key , chapter selection soft key [MACRO], and soft key [(OPRT)] in the stated order.
The following screen is displayed.

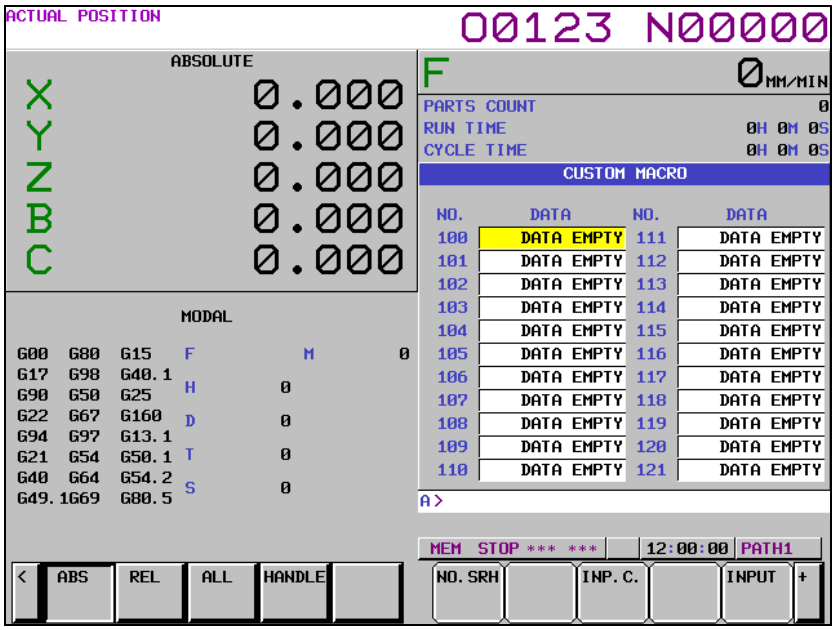


Fig. 12.3.6 (a) CUSTOM MACRO screen (10.4-inch display unit)

3 Move the cursor to the variable number to set using either of the following methods:
- Enter the variable number and press soft key [NO.SRH].
- Move the cursor to the variable number to set by pressing page keys PAGE and/or PAGE and cursor keys ↑, ↓, ←, and/or →.

4 Enter data with numeric keys and press soft key [INPUT].

5 To set a relative coordinate in a variable, press address key X, Y, or Z, then press soft key [INP.C.].

6 To set a blank in a variable, just press soft key [INPUT]. The value field for the variable becomes blank.

Explanation

If the value of a variable produced by an operation is not displayable, an indication below is provided.
When the significant number of digits is 12 (with bit 0 (F16) of parameter No. 6008 set to 0):

Variable value range	Variable value indication
0 < Variable value < +0.00000000001	+Underflow
0 > Variable value > -0.00000000001	-Underflow
Variable value > 999999999999	+Overflow
Variable value < -999999999999	-Overflow

When the significant number of digits is 8 (with bit 0 (F16) of parameter No. 6008 set to 1):

Variable value range	Variable value indication
0 < Variable value < +0.0000001	+Underflow
0 > Variable value > -0.0000001	-Underflow
Variable value > 99999999	+Overflow
Variable value < -99999999	-Overflow

12.3.7 Displaying and Setting Real Time Custom Macro Data

Real time macro variables (RTM variables) are dedicated to real time custom macros.
RTM variables are divided into temporary real time macro variables (temporary RTM variables) and permanent real time macro variables (permanent RTM variables).
The values of permanent RTM variables are kept stored after the power is turned off.
Temporary RTM variables are cleared to 0 when the power is turned off.
System variables (DI/DO variables) dedicated to real time custom macros are used to read and write PMC interface signals. Data can be read and written in bit and byte units. Before writing a signal, release the protection of the corresponding variable on the PMC signal protect screen.

Displaying and setting real time custom macro variables

Procedure

1 Press function key SYSTEM.

2 Press the continuous menu key ▷ several times, then press chapter selection soft key [R.TIME MACRO]. The following screen appears:

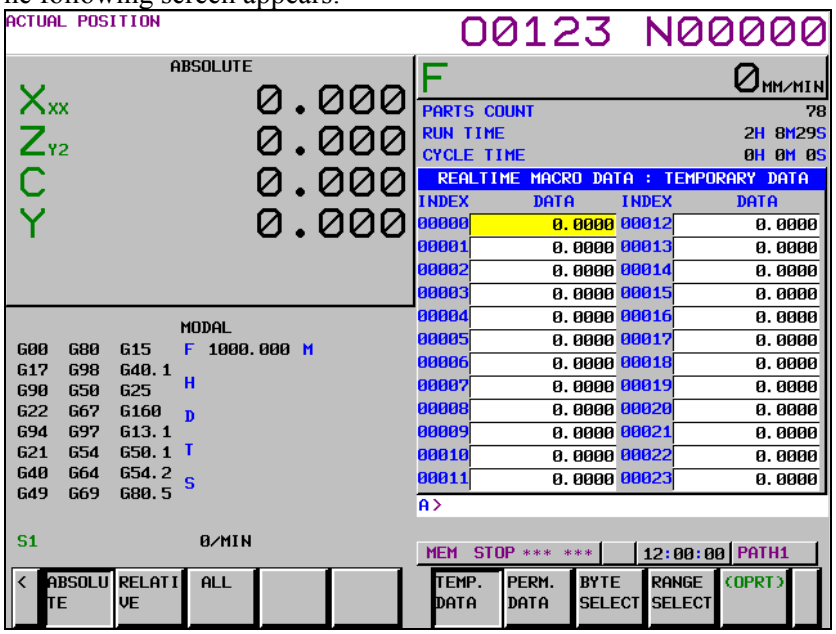


Fig. 12.3.7 (a) REAL TIME MACRO screen (10.4-inch display unit)

3 To display or set real time custom macro variables of which values are not stored at power-off, press soft key [TEMP. DATA].

4 To display or set real time custom macro variables of which values are stored at power-off, press soft key [PERM. DATA].

5 Move the cursor to the number of a real time custom macro variable you want to set using either of the following methods:

- Enter the number of a real time custom macro variable and press soft key [NO. SRH].
- Move the cursor to the number of a real time custom macro variable you want to set by pressing page keys PAGE ↑ and/or PAGE ↓ and cursor keys ↑, ↓, ←, and/or →.

6 Enter data.

Displaying and setting DI/DO variables

Procedure

For setting in byte units:

1 Press function key SYSTEM.

2 Press the continuous menu key ▷ several times, then press chapter selection soft key [R.TIME MACRO]. The following screen appears:

3 Press soft key [BYTE SELECT]. The following screen appears:

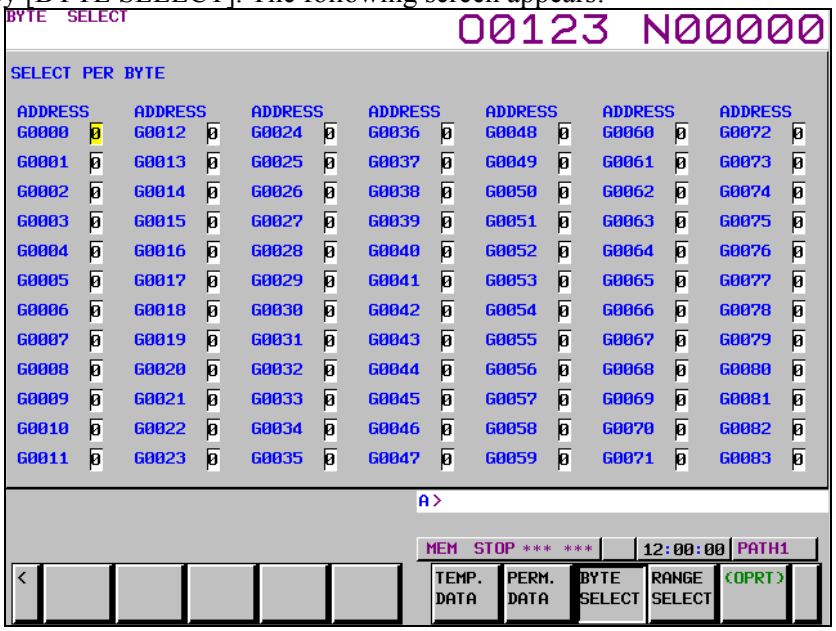


Fig. 12.3.7 (b) BYTE SELECT screen (10.4-inch display unit)

4 Move the cursor to the number of a DI/DO variable you want to set using either of the following methods:

- Enter the number and press soft key [NO. SRH].
- Move the cursor to a desired number by pressing page keys PAGE ↑ and/or PAGE ↓ and cursor keys ↑, ↓, ←, and/or →.

5 Enter data.

12.3.8 Displaying and Setting the Software Operator's Panel

Operations on the MDI unit can substitute for the functions of switches on the machine operator's panel. This means that a mode selection, jog feed override selection, and so forth can be made with operations on the MDI unit, eliminating the need to use the corresponding switches on the machine operator's panel. Jog feed can be performed using numeric keys.

Procedure for displaying and setting the software operator's panel

Procedure

1 Press function key OFS SET.

2 Press the continuous menu key ▷, then press chapter selection soft key [OPR].

3 The screen consists of several pages. Press page key PAGE ↑ or PAGE ↓ until the desired screen is displayed.

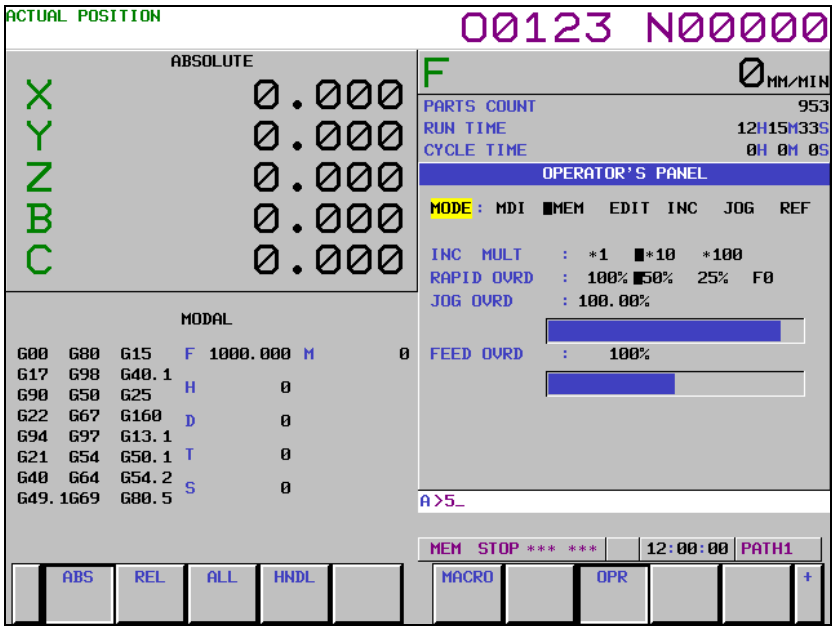


Fig. 12.3.8 (a) Page 1 of Software Operator's Panel screen (without the manual handle feed function) (10.4-inch display unit)

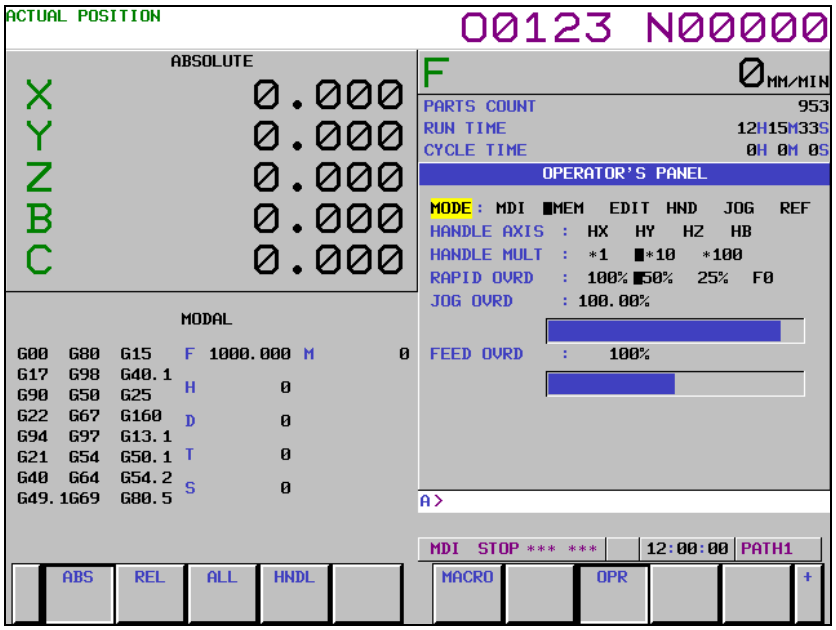


Fig. 12.3.8 (b) Page 1 of Software Operator's Panel screen (with the manual handle feed function) (10.4-inch display unit)

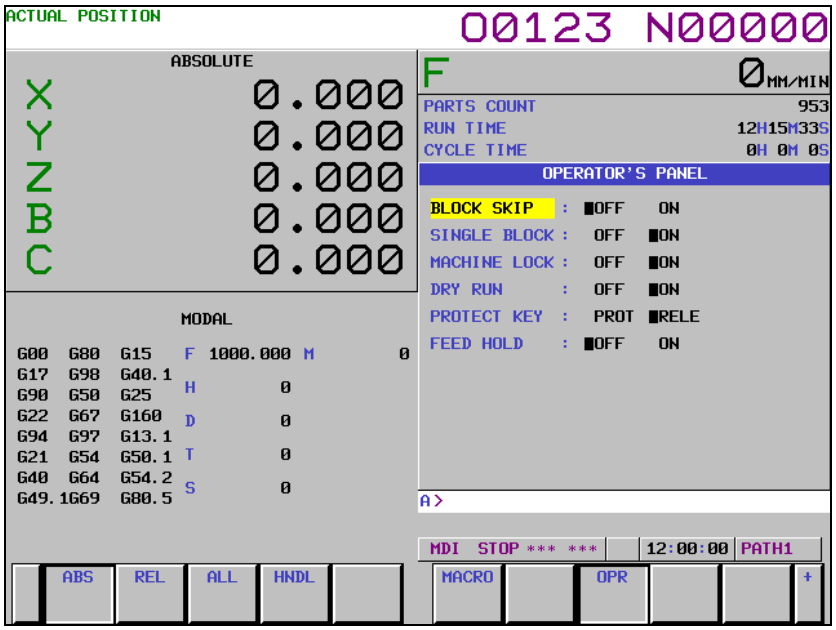


Fig. 12.3.8 (c) Page 2 of Software Operator's Panel screen (10.4-inch display unit)

4 Move the cursor to the desired switch by pressing cursor key ↑ or ↓.

5 Push the cursor key ← or → to match the mark ■ to an arbitrary position and set the desired condition.

6 Press one of the following arrow keys to perform jog feed. Press the JOG 5 key together with an arrow key to perform jog rapid traverse.

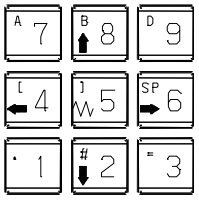


Fig. 12.3.8 (d) MDI arrow keys

Explanation

- Valid operations

The valid operations on the software operator's panel are shown below. Whether to use each group can be chosen using parameter No. 7200. Those groups that are not used are not displayed on the software operator's panel.

Group1 : Mode selection
Group2 : Selection of jog feed axis, Manual rapid traverse
Group3 : Selection of manual pulse generator feed axis, selection of manual pulse magnification
Group4 : Jog federate, federate override, rapid traverse override
Group5 : Optional block skip, single block, machine lock, dry run
Group6 : Protect key
Group7 : Feed hold

- Screens on which jog feed is valid

When the LCD indicates other than the software operator's panel screen and self-diagnosis screen, jog feed is not conducted even if the arrow key is pushed.

- **Jog feed and arrow keys**

The feed axis and direction corresponding to the arrow keys can be set with parameters Nos. 7210 to 7217.

- **General purpose switches**

For the meanings of these switches, refer to the manual issued by machine tool builder.

12.3.9 Setting and Displaying Tool Management Data

The tool management function totally manages tool information including tool offsets and tool life information.
This function provides a magazine screen and tool management screen. This subsection explains how to set these screens.

12.3.9.1 Displaying and setting magazine screen

1 Press function key .

2 Press chapter selection soft key [TOOL MANAGER].
Alternatively, press function key  several times until the tool management screen appears.
Press soft key [MAGAZINE].

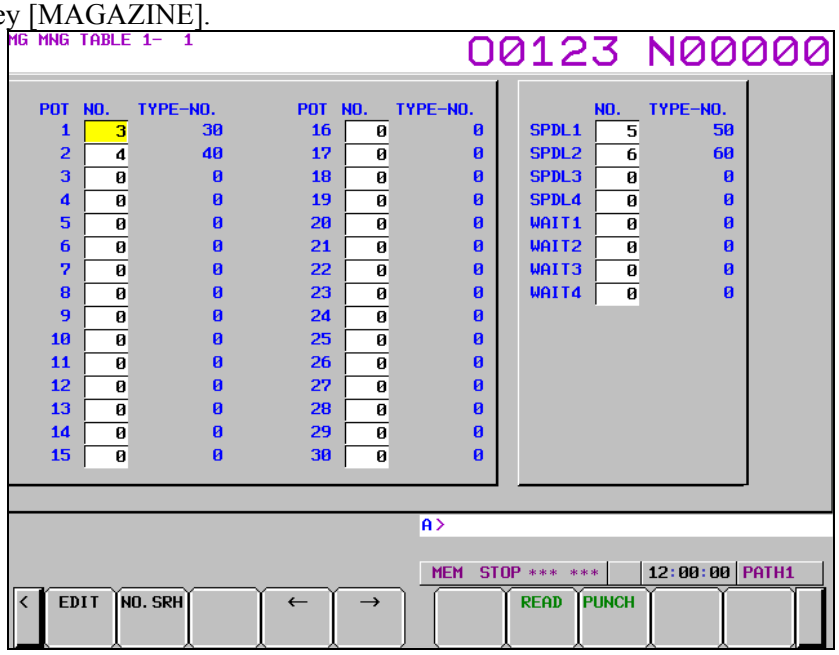


Fig. 12.3.9.1 (a) Tool management magazine screen (10.4-inch display unit)

3 By using the page keys, cursor keys, and soft keys [←] and [→], move the cursor to the pot No. position of the magazine for which you want to set or modify data.
Alternatively, type the number of the tool management data you wan to set or modify, and press soft key [NO. SRH].

4 Press soft key [EDIT].

5 To set the tool management data number of a pot, type the tool management data number, then press soft key [INPUT].
To delete the tool management data number set for a pot, follow the steps below.
<1> Press soft key [ERASE].
<2> Press soft key [CURDAT]. To delete the tool management data numbers registered for all pots, press soft key [ALL].
<3> Press soft key [EXEC]. Alternatively, type 0.

6 To end the edit operation, press soft key [EXIT].
This returns the screen display to the conventional tool management screen.

Explanation

- Another method

Magazine data can be input/output also by using external I/O devices.
See III-8, “DATA I/O”.

- Displayed data

POT : As many pots as specified in parameter No. 13222, 13227, 13232, or 13237 (settable for each magazine) are displayed. The starting pot number can be set for each magazine in parameter No. 13223, 13228, 13233, or 13238.

NO. : Tool management data numbers are displayed.
In the following cases, tools cannot be set in magazines:
1. A tool management data number beyond the range from 0 to (the number of valid pairs in parameter No. 13220) is set.
2. Tool management data is invalid (bit 1 of tool information is 0).
3. The tool management data number to be set is already set for another pot.

TYPE NO. : Tool type numbers corresponding to tool management data numbers are displayed.
Values cannot be set on this screen.

SPDL : The tool management data numbers and tool type numbers of spindle positions are displayed.

WAIT : The tool management data numbers and tool type numbers of wait positions are displayed.

- Edit mode

To edit data, press soft key [EDIT] to enter tool management data edit mode.

- Tool management data number

Tool management data numbers can optionally be extended to 64, 240, or 1000 pairs. In addition, the number of tools used can be set in parameter No. 13220.

- Display of the spindle positions and wait positions

The spindle position and wait position can be set for each path.

12.3.9.2 Displaying and setting tool management screen

Procedure

1 Press function key .

2 Press chapter selection soft key [TOOL MANAGER].
Alternatively, press function key  several times until the tool management screen appears.

3 Press soft key [TOOL MANAGER].

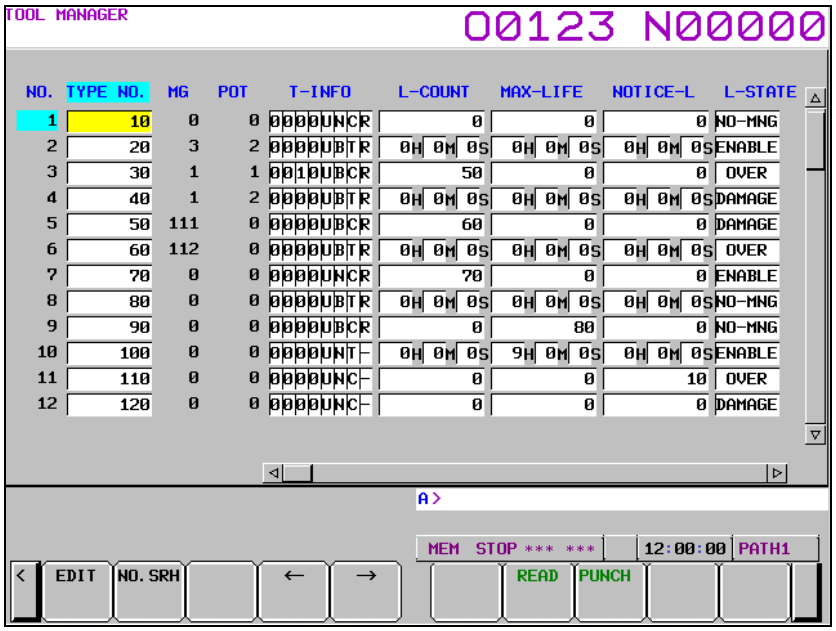


Fig. 12.3.9.2 (a) Tool management data screen (10.4-inch display unit)

4 By using the page keys, cursor keys, and soft keys [←] and [→], move the cursor to the position of the tool information of the tool number for which you want to set or modify data.
Alternatively, type the tool number of the data you want to set or modify, and press soft key [NO. SRH].

5 Press soft key [EDIT].

6 To set tool data, type a desired value, then press soft key [INPUT].
To delete tool information, follow the steps below.
<1> In step 4, move the cursor to the tool information you want to erase.
<2> Press soft key [ERASE].
<3> Press soft key [CURDAT]. To delete entire tool information, press soft key [ALL].
<4> Press soft key [EXEC].

7 To end the edit operation, press soft key [EXIT].
This returns the screen display to the conventional tool management screen.

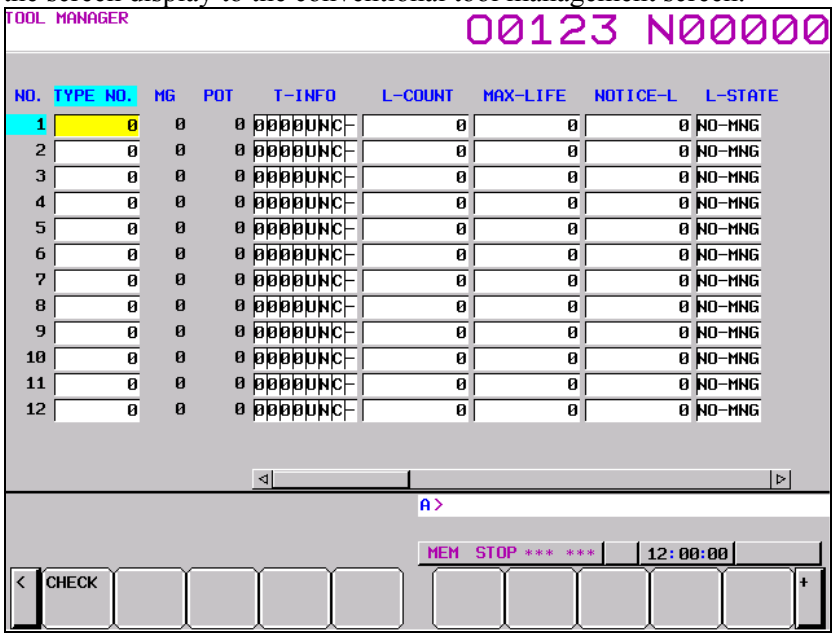


Fig. 12.3.9.2 (b) Tool management data screen (check function) (10.4-inch display unit)

8 When soft key [CHECK] is pressed, if there are tools with the same number but with different count types (count and time), the cursor moves to the tool type number of the smallest tool management number in the tool type numbers and a warning message appears.
The warning messages issued by the check function are shown below.
<1> For the same type of tools with different count types
L-COUNT TYPE MISMATCH:XXXXXXXX
<2> For the same type of tools with the same count type
L-COUNT TYPE MATCH

Explanation

- Another method

Tool management data, customize data, and names set for tool states can be input/output also by using external I/O devices.
See III-8, “DATA I/O”.

- Displayed information

• Life information

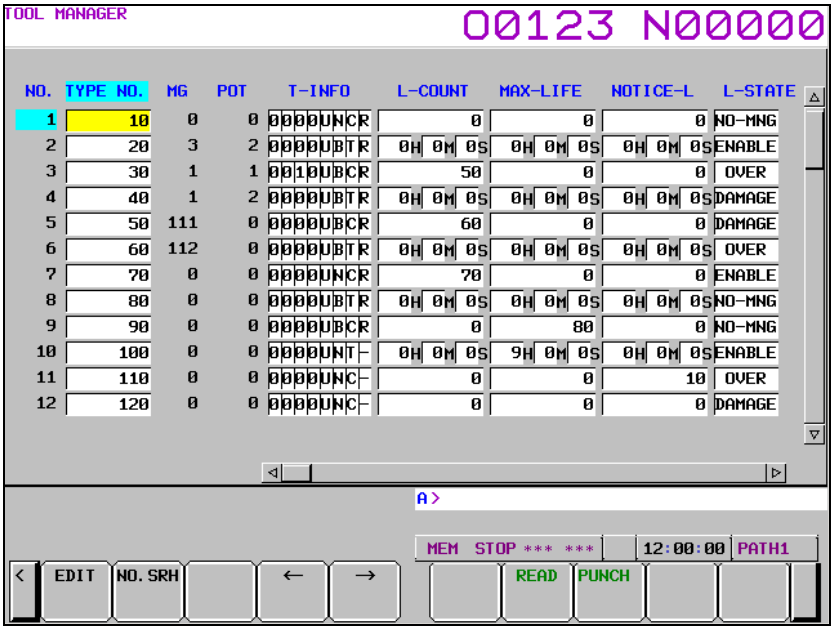


Fig. 12.3.9.2 (c) Tool management data life status screen(10.4-inch display unit)

NO. : Tool management data numbers are displayed. These numbers can be displayed but cannot be set. The tool management data number of edited data is kept blinking until tool management data edit mode ends. Tool management data numbers set or deleted on the magazine screen also blink on the tool management screen.

TYPE NO. : Tool type numbers are displayed.
A value from 0 to 99,999,999 can be set.

MG : The magazine number assigned to each tool is indicated. These numbers can be displayed but cannot be set.

POT : The pot number assigned to each tool is indicated. These numbers can be displayed but cannot be set.

T-INFO : The following five types of information are displayed sequentially from the right:
- Tool management data: Valid (R)/invalid (-))
- Life count type: Time (T)/count (C)
- Tool type: Oversize tool (B)/normal tool (N)
- Data access: Locked (L)/unlocked (U)
- When a tool is not under life management:
Included in the tools to be searched (1)/not included in the tools to be searched (0)

NOTE
1 The tool types and data access information vary depending on the specifications defined by the machine tool builder.
2 The same type of tools must have the same life count type.

L-COUNT : The number of use times/use period of time of each tool is indicated.
Up to 99,999,999 times or 999 hours 59 minutes 59 seconds can be set.
MAX-LIFE : The maximum life value of each tool is indicated.
Up to 99,999,999 times or 999 hours 59 minutes 59 seconds can be set.
NOTICE-L : Noticed life value of a tool
Up to 99,999,999 times or 999 hours 59 minutes 59 seconds can be set.
L-STATE : Current tool state
One of the four states, including invalid (0), present (1, 2), not present (3), and broken (4), is indicated. The numbers in parentheses are data values used when these states are input in MDI.

- Spindle speed/feedrate

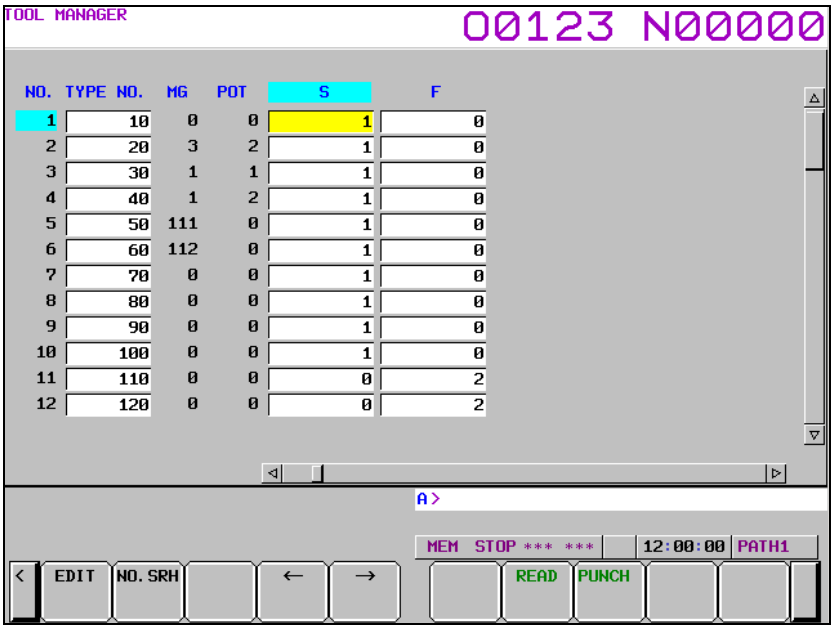


Fig. 12.3.9.2 (d) Tool management data spindle speed/feedrate screen (10.4-inch display unit)

S : Spindle speed.
A value from 0 to 99,999 can be set.
F : Feedrate.
A value from 0 to 99,999,999 can be set.

- Tool offset information

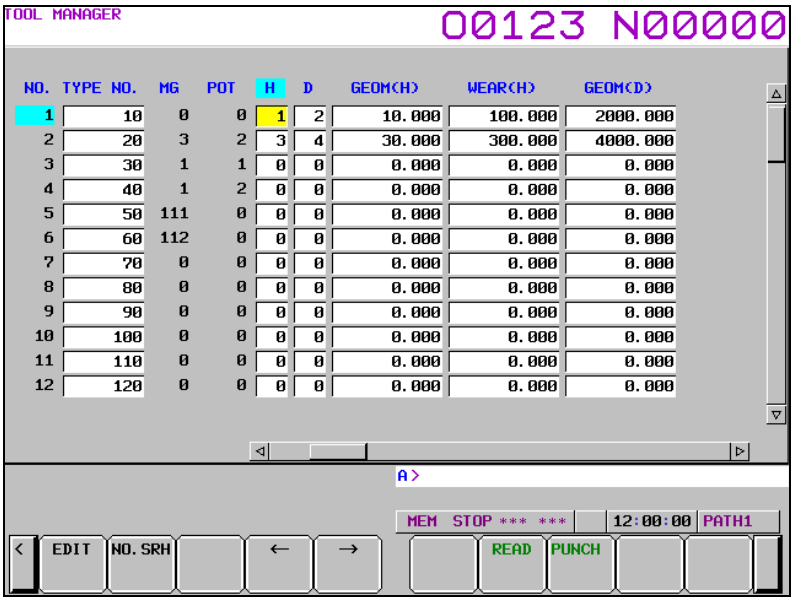


Fig. 12.3.9.2 (e) Tool management data tool offset screen (10.4-inch display unit)

H : Tool length compensation number (for machining center systems only).
A value from 0 to 999 can be set.
D : Cutter compensation number (for machining center systems only).
A value from 0 to 999 can be set.
TG : Tool geometry compensation number (for lathe systems only).
A value from 0 to 999 can be set.
TW : Tool wear compensation number (for lathe systems only).
A value from 0 to 999 can be set.

The displayed offset data is determined depending on the option configuration and parameter settings (bits 1(DCR), 2(DOY), 3(DOB), 4(DO2), 6(DOT), and 7(DOM) of parameter No. 13202).

- Customize information

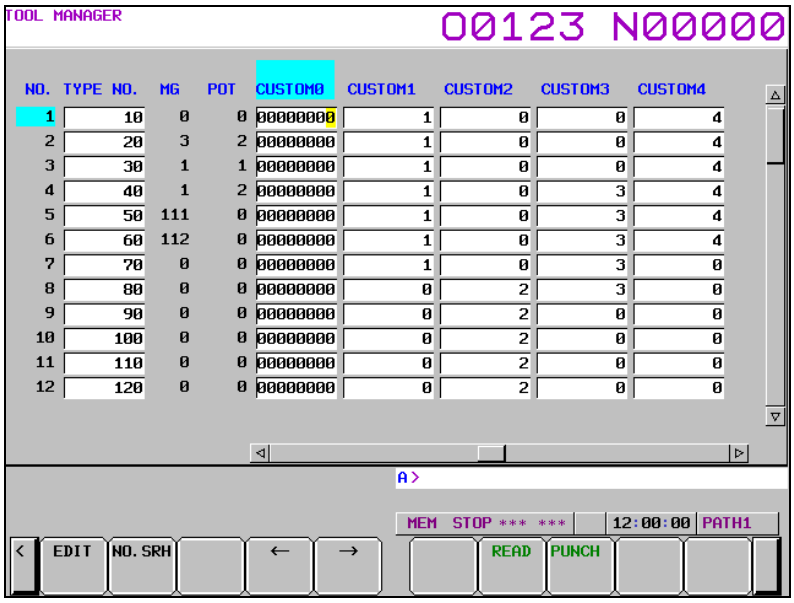


Fig. 12.3.9.2 (f) Tool management data customize data screen (10.4-inch display unit)

CUSTOM0 : Bit-type customize information.
For each bit, 1 or 0 can be input.

CUSTOM1 to 4 : Customize information. Any value from -99,999,999 to 99,999,999 can be set.
CUSTOM5 to 20 : Customize information. These items are displayed only when customize data extension option (5 to 20) of the tool management function is enabled. Any value from -99,999,999 to 99,999,999 can be set.
CUSTOM21 to 40 : Customize information. These items are displayed only when customize data extension option (5 to 40) of the tool management function is enabled. Any value from -99,999,999 to 99,999,999 can be set.

- Tool management data number

Tool management data numbers can optionally be extended to 64, 240, or 1000 pairs. The number of tools used can be set in parameter No. 13220.

- Edit mode

To edit data, press soft key [EDIT] to enter tool management data edit mode.

- Life count override

When the period of time is set as the tool life count type, the life count can be overridden if bit 2 (LFV) of parameter No. 6801 is set to 1. Specify an override value by using a switch on the machine operator's panel.

Example)

When cutting is performed for 10 minutes with an override of 0.1, one minute is counted in the tool life counter.

- Tool management extension function

When tool management extension functions are enabled, you can use the following functions in addition to the tool management functions:

- A value with a decimal point can be set as customize data.
  The maximum number of decimal places can be set to 7 by G10 data input and file input.
- Various types of tool management data can be protected using KEY signals.
  An attempt to enter a value to a protected data causes a warning message "WRITE PROTECT."
- Tool life count period selection

Bit 5 of tool information is used to make a life count period selection.

Item		Description
Data length		1 byte (flag data)
#5	REV	0: A life count period of 1 sec is used. 1: A life count period of 8 msec is used.

Range of count is as follows.

1sec : 0 to 3,599,999 seconds (999 hours 59 minutes 59 seconds)
8msec : 0 to 3,599,992 ms (59 minutes 59 seconds 992 milliseconds)

12.3.9.3 Each tool data screen

Each tool data screen

Procedure

1 Press function key .

2 Press chapter selection soft key [TOOL MANAGER].
Alternatively, press function key  several times until the tool management screen appears.

3 Press soft key [EACH TOOL]. The each tool data screen appears.

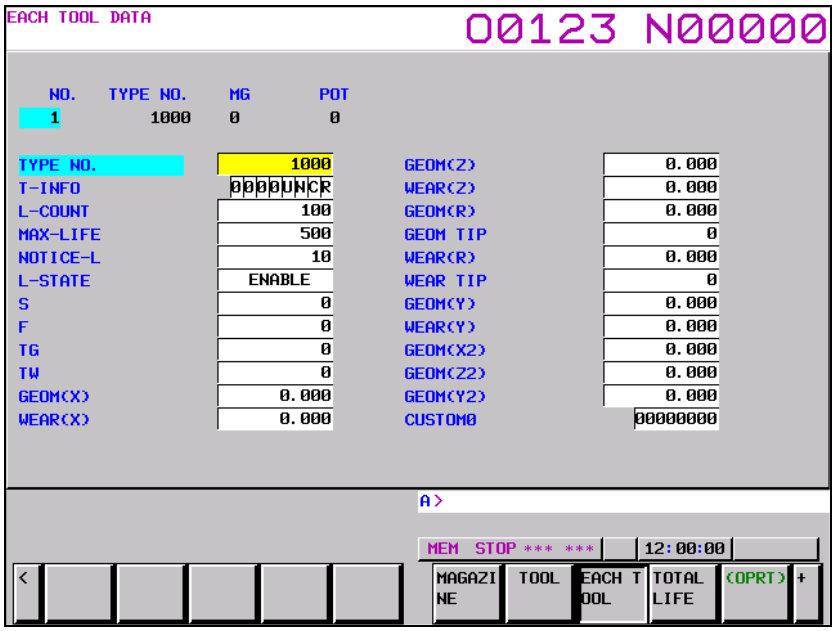


Fig. 12.3.9.3 (a) Each tool data screen (10.4-inch display unit)

Explanation

- Header

The following four data items are displayed: NO., TYPE NO., MG, and POT.
When the data table of a tool extends over two or more pages, the same header is displayed on these pages.

- Data table

The data table shows data items related to a tool at a time.
The data items are displayed from the upper left to the lower left, the upper right, and the lower right in ascending order of screen display position numbers specified using the customize function. The number of digits displayed for one data item is fixed to 11.
Up to 24 data items are displayed on one page. (12 data items × 2 columns)
When more than 24 data items are set for a tool, the 25th and subsequent data items are displayed on the next page. (Up to three pages)
When a data item is set as a screen element of the tool management data screen twice or more using the tool management data display customize function (one of the tool management extension functions), only the data item with the smaller display position number is displayed. When no value is set for a data item, the field for the data item is not displayed and the next data item is displayed.

Key operations

- MDI key operations

Numeral keys Inputs a numeric value.

PAGE ↑ Displays the previous page for the same tool.

PAGE ↓ Displays the next page for the same tool.

↑ Moves the cursor to the previous data item.

When the cursor is on the top row of the data table, it moves to the bottom row of the left column. When the cursor is on the first data item, it moves to the last data item.

↓ Moves the cursor to the next data item.

When the cursor is on the bottom row of the data table, it moves to the top row of the right column. When the cursor is on the last data item, it moves to the first data item.

Moves the cursor left on the screen.

When the cursor is on the left column of the data table, it moves to the right column on the row immediately above. When the cursor is on the first data item, it moves to the last data item.

Moves the cursor right on the screen.

When the cursor is on the right column of the data table, it moves to the left column on the row immediately below. When the cursor is on the last data item, it moves to the first data item.

- Soft key operations

Soft key [EDIT]

Enters the management data edit mode.

Soft key [NO.SRH]

Displays data with a tool management data number when pressed after the tool management data number is entered.

Soft key [PREV.TOOL]

Returns to the previous tool management data number.

Soft key [NEXT.TOOL]

Proceeds to the next tool management number.

Operation in the management data edit mode

To edit data, press soft key [EDIT] to enter the management data edit mode.

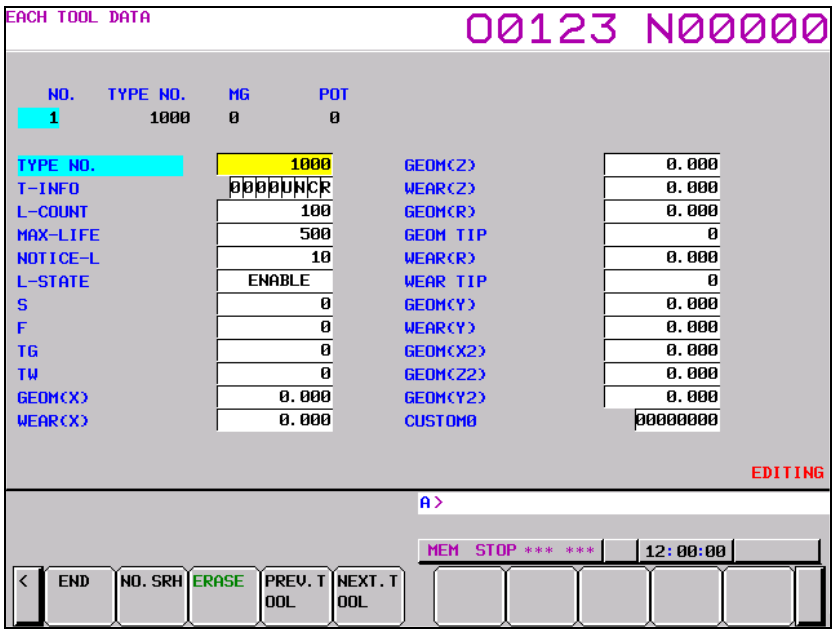


Fig. 12.3.9.3 (b) Each tool data screen (10.4-inch display unit)

In the management data edit mode, “EDITING” is displayed at the lower right of the screen.
In addition to the above key operations, the following key operations are available in the management data edit mode:

INPUT Actually writes a value entered using numeric keys.

Soft key [END]

Ends the management data edit mode.

Operations for editing tool information are the same as those on the tool management data screen.

12.3.9.4 Displaying the total life of tools of the same type

Total life data screen

Procedure

1 Press function key .

2 Press chapter selection soft key [TOOL MANAGER].

Alternatively, press key  several times until the tool management screen appears.

3 Press soft key [TOTAL LIFE]. The total life data screen appears.

4 Press soft key [(OPRT)], then soft key [CHANGE] to switch the display between total life data of the count counting type tools and that of the time counting type tools.

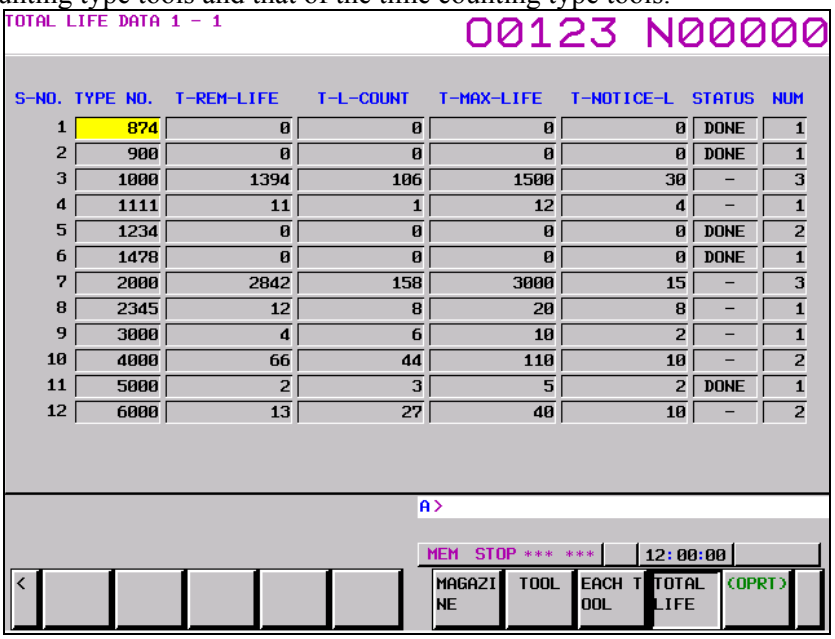


Fig. 12.3.9.4 (a) Count display (10.4-inch display unit)

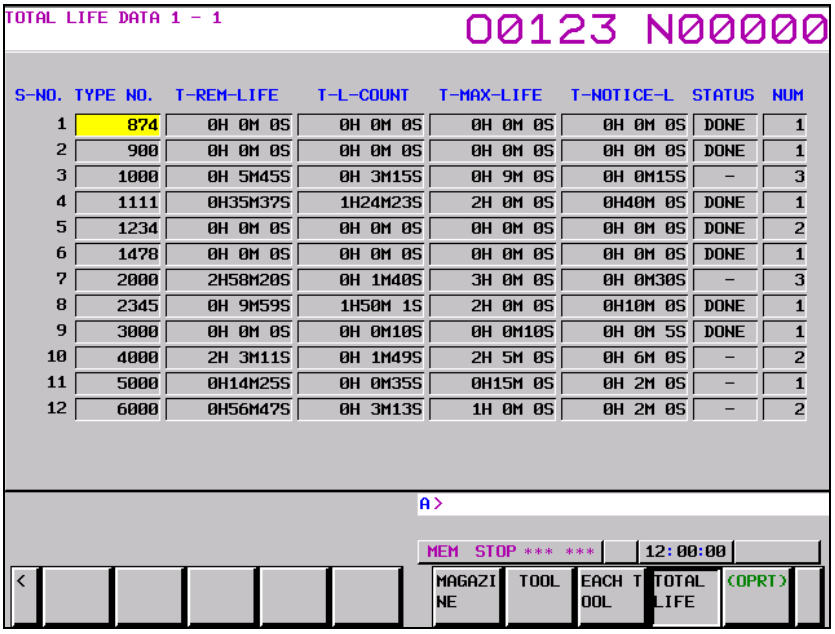


Fig. 12.3.9.4 (b) Time display (10.4-inch display unit)

Displayed information

S-NO. : Sequential number of each tool type
TYPE NO. : Tool type number
T-REM-LIFE : Total of remaining life values of tools with the same tool type number
T-L-COUNT : Total of used counts/times of tools with the same tool type number
T-MAX-LIFE : Total of maximum life values of tools with the same tool type number
NUM : Number of tools with the same tool type number
T-NOTICE-L : Total of notice life values of tools with the same tool type number when the tool life arrival signal is to be output for each tool type
STATUS : Whether the tool life arrival signal has been output when the signal is to be output for each tool type
Either of the two states (UNDONE and DONE) is displayed.

When bit 3 (ETE) of parameter No. 13200 is set to 0 and bit 2 (TRT) of parameter No. 13200 is set to 1, the tool life arrival signal is to be output for each tool type number and T-NOTICE-L and STATUS are displayed.

Key operations

- MDI key operations



Displays the previous page. The cursor moves to the last data item on that page.



Displays the next page. The cursor moves to the first data item on that page.


Moves the cursor up on the screen.


Moves the cursor down on the screen.


Disabled.


Disabled.

- Soft key operations

Soft key [DETAIL]
Displays the detailed life data screen.
Soft key [UPDATE]
Updates data displayed on the total life data screen.
Soft key [CHANGE]
Switch the display between the count and time types.
Soft key [T-TYPENO. SRH]
Moves the cursor to a tool type number when pressed after the number is entered.
Soft key [T-ASCE -SORT]
Sorts data in ascending order of tool type numbers.
Soft key [T-DESC -SORT]
Sorts data in descending order of tool type numbers.
Soft key [R-ASCE -SORT]
Sorts data in ascending order of remaining life values.
Soft key [R-DESC -SORT]
Sorts data in descending order of remaining life values.

NOTE
1 After soft key [T-ASCE -SORT], [T-DESC -SORT], [R-ASCE -SORT], or [R-DESC -SORT] is pressed, the cursor is positioned at the top of page 1 of the total life data screen.

NOTE
2 When the power is turned on, data of the count counting type is displayed in ascending order of tool type numbers. When the display type is changed or data is sorted in a different order, the status is kept.
3 If soft key [DETAIL] is pressed when no data is displayed on the total life data screen, the warning message “NO DETAILED LIFE DATA SCREEN” is output.
4 If an unregistered tool type is specified and soft key [T-TYPENO. SRH] is pressed, the warning message “UNREGISTERED NUMBER” is output.

Detailed life data screen

Procedure

1 Press function key .
2 Press chapter selection soft key [TOOL MANAGER].
Alternatively, press  several times until the tool management screen appears.
3 Press soft key [TOTAL LIFE]. The total life data screen appears.
4 Press soft key [(OPRT)], then soft key [DETAIL]. The detailed life data screen appears.

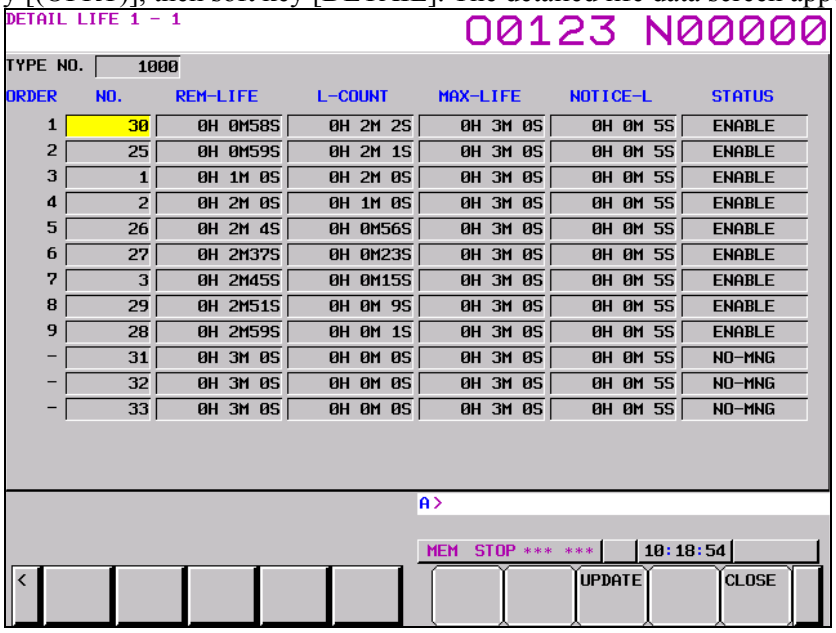


Fig. 12.3.9.4 (c) Detailed life data screen (10.4-inch display unit)

- Displayed information

TYPE NO. : Tool type number
ORDER : Sequential number in ascending order of remaining life times or the order in which the customize data is set.
NO. : Tool management data number
REM-LIFE : Remaining life value obtained by subtracting the life count value from the maximum life value
L-COUNT : Total used count or time of the tool
MAX-LIFE : Maximum life value of the tool
NOTICE-L : Notice life value of the tool
STATUS : Life status of the tool
One of the four states (NO-MNG (0), ENABLE (1, 2), NONE (3), and DAMAGED (4)) is displayed.

Key operations

- MDI key operations



Displays the previous page.



Displays the next page.


Moves the cursor up on the screen. The cursor moves to the last data item on that page.


Moves the cursor down on the screen. The cursor moves to the first data item on that page.


Disabled.


Disabled.

- Soft key operations

Soft key [UPDATA]

Updates data displayed on the detailed life data screen.
The cursor is positioned at the beginning of page 1 after this soft key is pressed.

Soft key [CLOSE]

Closes the detailed life data screen and returns to the total life data screen.

NOTE
1 When soft key [CLOSE] is pressed and the total life data screen is displayed again, the cursor on the total life screen data is positioned as follows.
2 When the tools with the tool type number displayed when the detailed data screen is closed are registered as tool management data, the cursor on the total life screen is positioned at that tool type number.
3 When the tools with the tool type number displayed when the detailed data screen is closed are not registered as tool management data, the cursor on the total life data screen is positioned at the first total life data.

12.3.9.5 Tool geometry data screen

Tool geometry data screen

Procedure

1 Press function key 

.

2 Press chapter selection soft key [TOOL MANAGER].
Alternatively, press 

several times until the tool management screen appears.

3 Press continuous menu key , then soft key [TOOL-GEOM]. The tool geometry data screen appears.

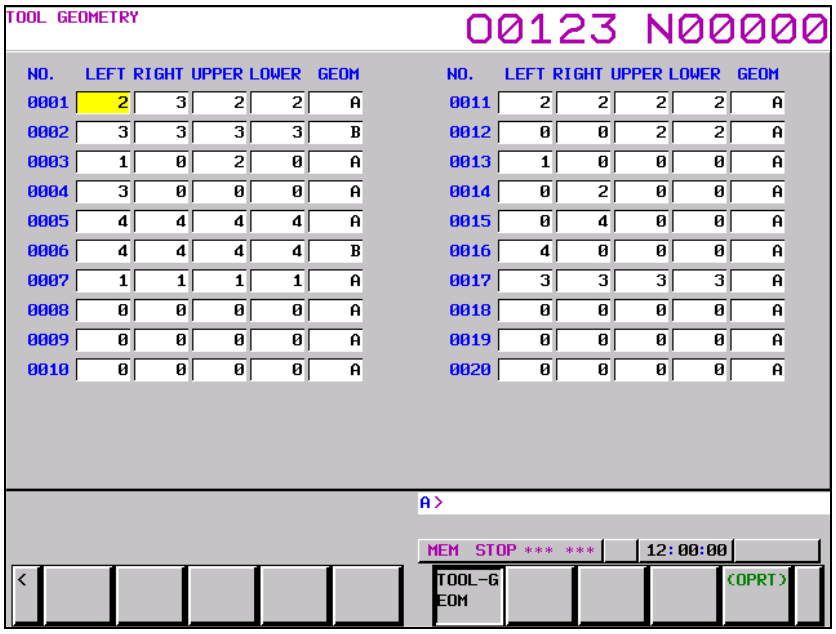


Fig. 12.3.9.5 (a) Tool geometry data screen (10.4-inch display unit)

- Displayed item

NO. : Tool geometry number
Up to 20 numbers can be displayed.

LEFT : Sets the number of pots on the left of the reference pot that are to be occupied.
A value between 0 and 4 can be set.

RIGHT : Sets the number of pots on the right of the reference pot that are to be occupied.
A value between 0 and 4 can be set.

UPPER : Sets the number of pots above the reference pot that are to be occupied.
A value between 0 and 4 can be set. (Use this item when the magazine is of the matrix type.)

LOWER : Sets the number of pots under the reference pot that are to be occupied.
A value between 0 and 4 can be set. (Use this item when the magazine is of the matrix type.)

Key operations

- Operations in the standard mode

MDI key operations

Numeral keys Inputs a numeric value.

Moves the cursor up on the screen.

Moves the cursor down on the screen.

Moves the cursor left on the screen.

Moves the cursor right on the screen.

Soft key operations

Soft key [NO.SRH]
Moves the cursor on a tool geometry number when pressed after the number is entered.

Soft key [EDIT]
Enters the data edit mode.

Soft key [F INPUT]
Inputs data related to the tool management functions.
This key is available only in the standard mode.
Put the NC in the EDIT mode.

Soft key [F OUTPUT]

Outputs data related to the tool management functions.

This key is available only in the standard mode.

Put the NC in the EDIT mode.

In the management data edit mode, the following key operations are available in addition to the above key operations.

- Operations in the edit mode

To edit data, press soft key [EDIT] to enter the edit mode. In the edit mode, “EDITING” is displayed at the lower right of the screen.

MDI key operation



Actually writes a value entered using numeric keys.

Soft key operation

Soft key [END]

Ends the data edit mode.

Example

Set the edit mode. When the tool geometry with tool geometry number 1 occupies 1 pot in the left direction, 0.5 pots in the right direction, and 1.5 pots in the down direction, set data as shown in the figure below:

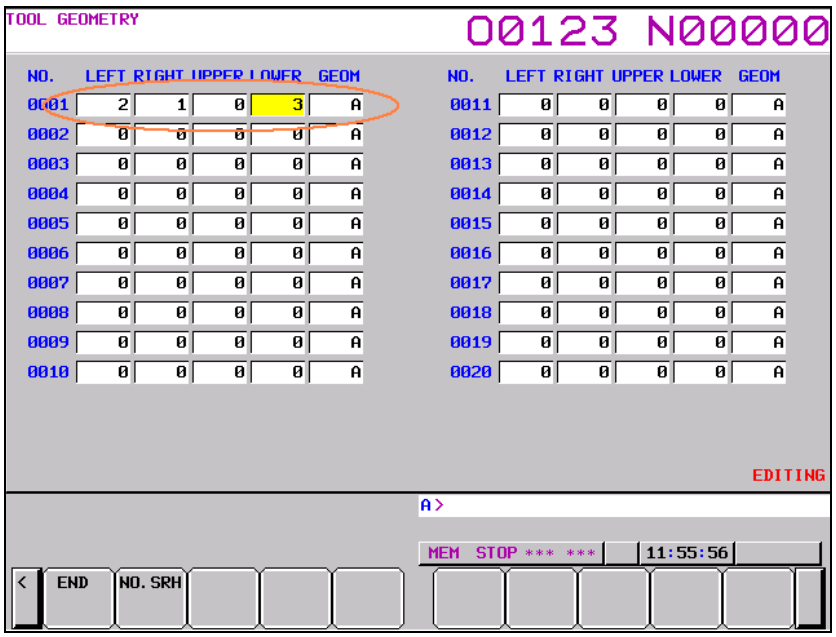


Fig. 12.3.9.5 (b) Example of setting data on the tool geometry data screen (10.4-inch display unit)

- Display of occupied pots in the magazine management table

Each pot occupied by a tool stored in another pot is indicated with an asterisk (*).

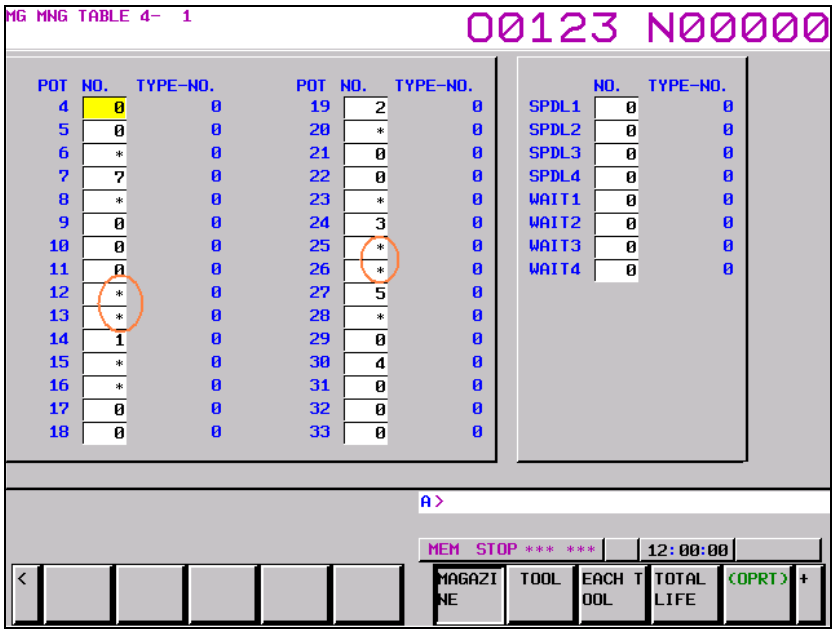


Fig. 12.3.9.5 (c) Magazine management table (10.4-inch display unit)

If a tool to be registered for a magazine is determined to interfere with another tool, the warning message "TOOL INTERFERENCE CHECK ERROR:xxxx,xxxx" is displayed. xxxx indicates the tool number of each of the two tools. If a tool is determined to interfere with multiple tools, only the first tool detected by the NC interference check is displayed.
If a tool is determined to interfere with the frame of a magazine, the tool number of the tool to be stored and "FRAME" are displayed.

- Search for an empty pot for a oversize tool

In the management data edit mode, pressing soft key [(OPRT)], then continuous menu key ▷ displays the soft keys for searching for an empty pot for a oversize tool.

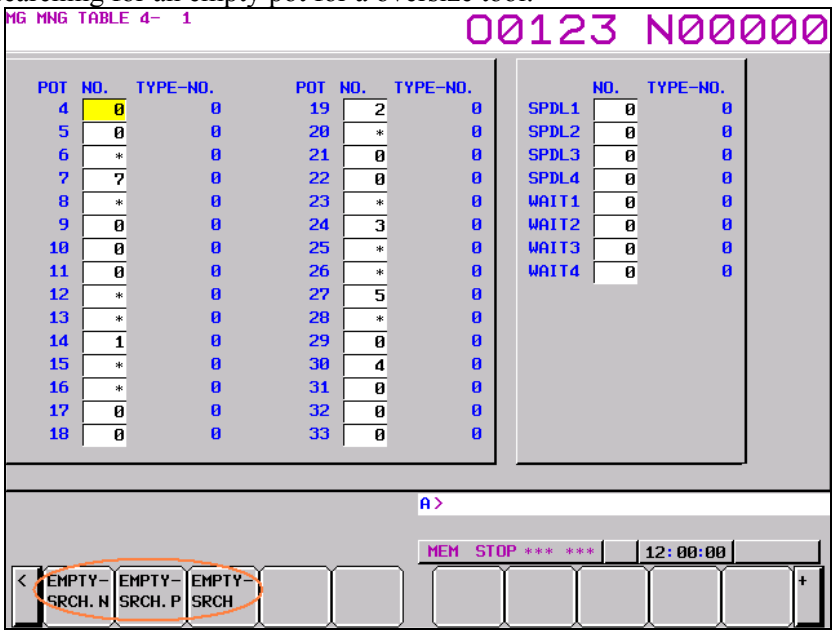


Fig. 12.3.9.5 (d) Searching for an empty pot for a oversize tool (10.4-inch display unit)

Enter the tool geometry number in the key-in buffer and press a search soft key. The cursor moves to an empty pot fit for the geometry.
EMPTY-SRCH.N : Searches for an empty pot forward.
EMPTY-SRCH.P : Searches for an empty pot backward.

EMPTY-SRCH : Searches for the pot nearest to the current position.

- Tool management screen

You can use bit 2 of tool information to switch between a oversize tool and normal tool. For a oversize tool, set a tool geometry number fit for the tool.

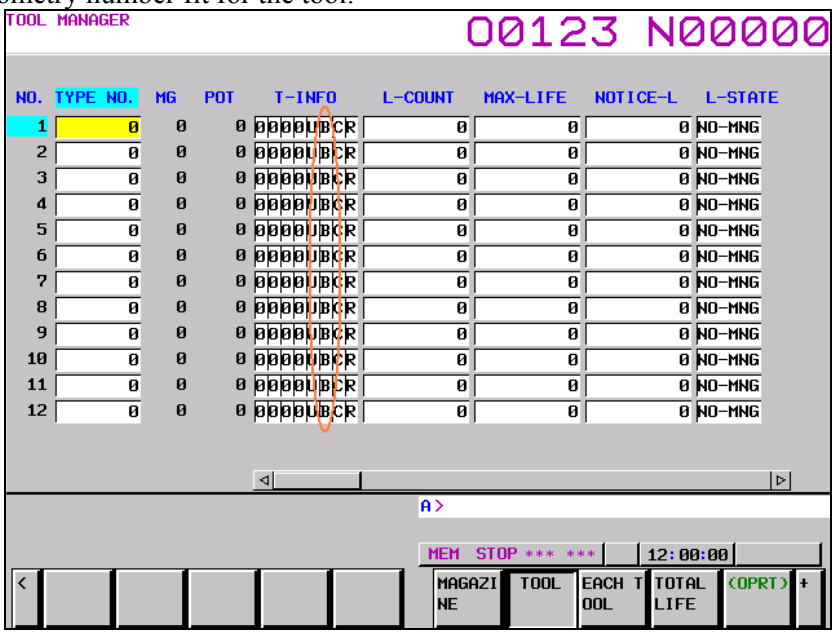


Fig. 12.3.9.5 (e) Bit for switching between a normal tool and oversize tool (10.4-inch display unit)

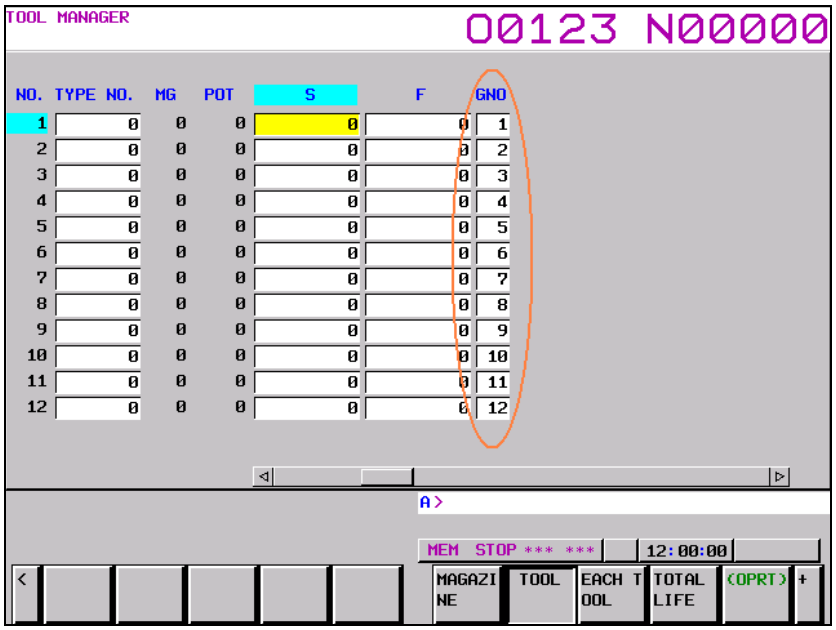


Fig. 12.3.9.5 (f) Tool geometry number (10.4-inch display unit)

12.3.10 Displaying and Switching the Display Language

The language used for display can be switched to another language.
A display language can be set using a parameter. However, by modifying the setting of the display language on this screen, the display language can be switched without turning off then on the power.

Displaying and setting the display language

Procedure

1 Press function key .

2 Press the continuous menu key  several times.

3 Press soft key [LANGUAGE] to display the language screen.

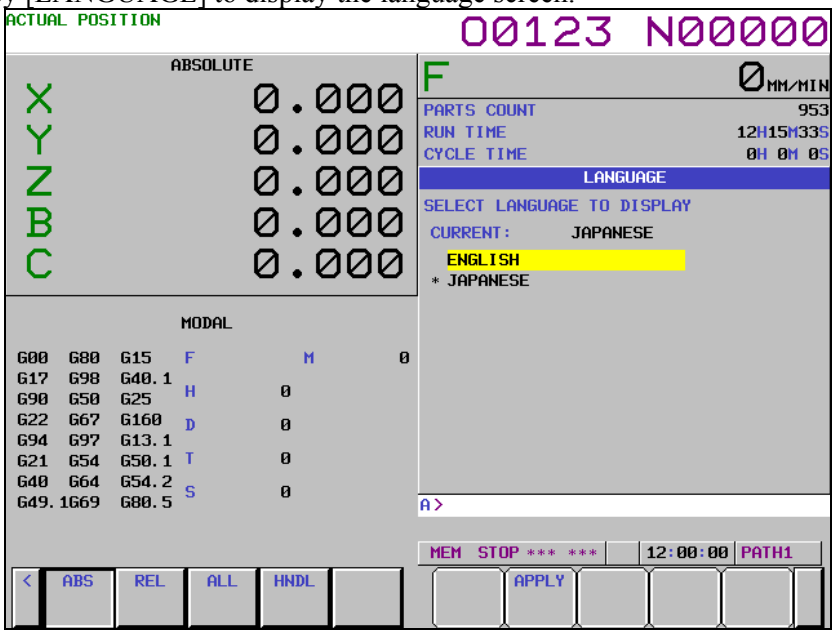


Fig. 12.3.10 (a) LANGUAGE screen (10.4-inch display unit)

4 Press page key  or , then press cursor keys ,  to move the cursor to a desired display language.

5 Press operation soft key [APPLY]. The display language is switched to the selected language. The language specified on this screen continues to be used if the power is turned off then back on.

Explanation

- Language switching

The language screen can be displayed if bit 0 (NLC) of parameter No. 3280 is set to 0.

- Selectable languages

The display languages selectable on this screen are as follows:

1. English
2. Japanese
3. German
4. French
5. Chinese (Traditional)
6. Chinese (Simplified)
7. Italian
8. Korean
9. Spanish
10. Dutch
11. Danish
12. Portuguese
13. Polish
14. Hungarian
15. Swedish
16. Czech

17. Russian
18. Turkish
19. Bulgarian
20. Rumanian

Among the languages listed above, English and other usable languages are displayed on the screen as a list of switchable languages.

Limitation

- Language parameter modification on the parameter screen

Which language to use for display is specified with parameter No. 3281. This parameter can be modified using the parameter screen as well. However, if a modification is made on the parameter screen, the new setting is not reflected until “APPLY” operation is performed on the language screen or the power is turned on again. If an invalid value is set in parameter No. 3281 on the parameter screen, the screen is displayed in English after the power is turned on again.

12.3.11 Protection of Data at Eight Levels

You can set eight CNC and PMC operation levels and one of eight protection levels for each type of CNC and PMC data.
When an attempt is made to change CNC and PMC data or output it to an external unit, the operation level is compared with the protection level to determine whether to allow the change or external output.

12.3.11.1 Operation level setting

You can set eight CNC and PMC operation levels.

Displaying and setting the operation level setting screen

Procedure

1 Press function key .

2 Press the continuous menu key  several times until soft key [PROTECT] is displayed.

3 Press soft key [PROTECT].
The operation level setting screen shown below is displayed.

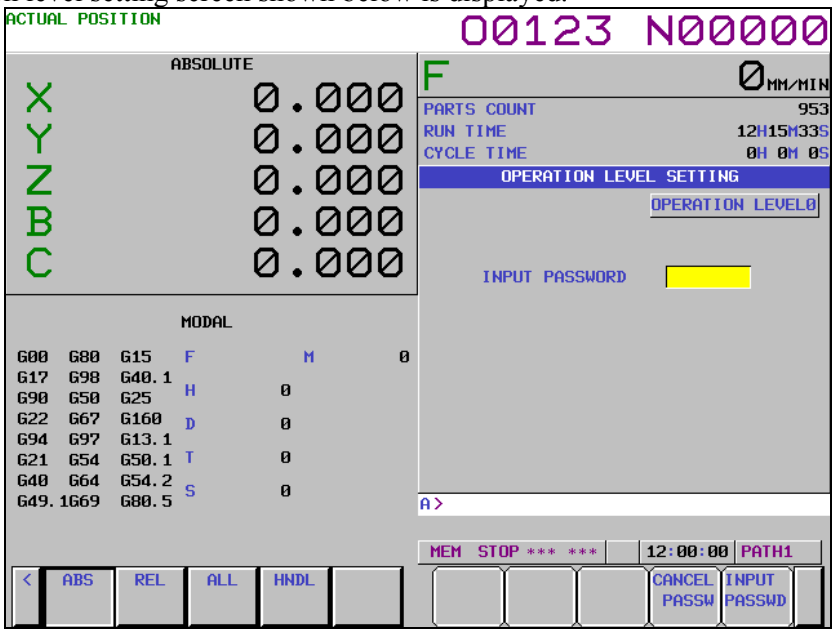


Fig. 12.3.11.1 (a) Operation level setting screen (10.4-inch display unit)

4 Key in the password for an operation level to be set/modified, then press soft key [INPUT PASSWD].
5 To return the operation level to 0, 1, 2, or 3, press soft key [CANCEL PASSWD].

Explanation

- Operation level setting

To select operation level 0 to 3, use the corresponding memory protection key signal.
To select operation level 4 to 7, use the corresponding password.

Table 12.3.11.1 (a) Operation level setting

Operation level	Setting	Sample grouping
7 (high)	Password	-
6	Password	MTB
5	Password	Dealer and integrator
4	Password	End user
3	Memory protection key signal	User level (level 1)
2	Memory protection key signal	User level (level 2)
1	Memory protection key signal	User level (level 3)
0 (low)	Memory protection key signal	User level (level 4)

When operation level 4 to 7 is set, the operation level remains unchanged until the password is cleared. (The operation level also remains unchanged if the power is turned off.)
Operation level 7 is reserved for CNC and PMC maintenance.

NOTE
When a password is being entered, an asterisk (*) is displayed instead of each entered character.

12.3.11.2 Password modification

The current operation level is displayed.
The password for each of operation levels 4 to 7 can be modified.

Displaying and setting the password modification screen

Procedure

1 Press function key .
2 Press the continuous menu key several times until soft key [PROTECT] is displayed.
3 Press soft key [PROTECT].
4 Press soft key [PASSWORD].
The PASSWORD CHANGE screen shown below is displayed.

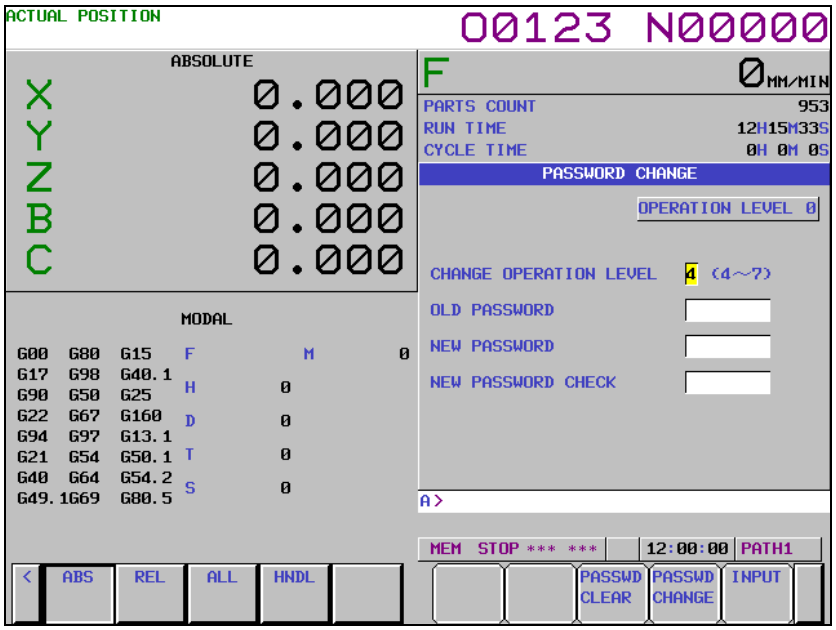


Fig. 12.3.11.2 (a) PASSWORD CHANGE screen (10.4-inch display unit)

5 Key in an operation level whose password is to be modified, then press soft key [INPUT].
6 Key in the current password for the operation level whose password is to be modified, then press soft key [INPUT].
7 Key in a new password, then press soft key [INPUT].
8 Key in the new password again for confirmation, then press soft key [INPUT].
9 Press soft key [PASSWD CHANGE].
10 To clear the password, press soft key [PASSWD CLEAR].

Explanation

Up to eight characters (only uppercase alphabetic characters and numeric characters) can be input.

NOTE

1 For a password, consisting of three to eight characters, the following characters are available:
- Uppercase alphabetic characters
- Numeric characters

2 When a password is being entered, an asterisk (*) is displayed instead of each entered character.
3 Whether a password can be changed at the current operation level is determined as follows:
- Password of an operation level higher than the current operation level
  Cannot be changed.
- Password of the current operation level
  Can be changed.
- Password of an operation level lower than the current operation level
  Can be changed (only to the initial password).

4 The set password is not displayed.
Be careful not to forget the password.

12.3.11.3 Protection level setting

The current operation level is displayed.
The change protection level and output protection level of each data item are displayed.
The change protection level and output protection level of each data item can be changed.

Confirmation based on protection level setting

Procedure

1 Press function key .

2 Press the continuous menu key  several times until soft key [PROTECT] is displayed.

3 Press soft key [PROTECT].

4 Press soft key [DATA LEVEL] to change the protection level of CNC data or press soft key [PMC LEVEL] to change the protection level of PMC data.
The following protection level change screen is displayed.

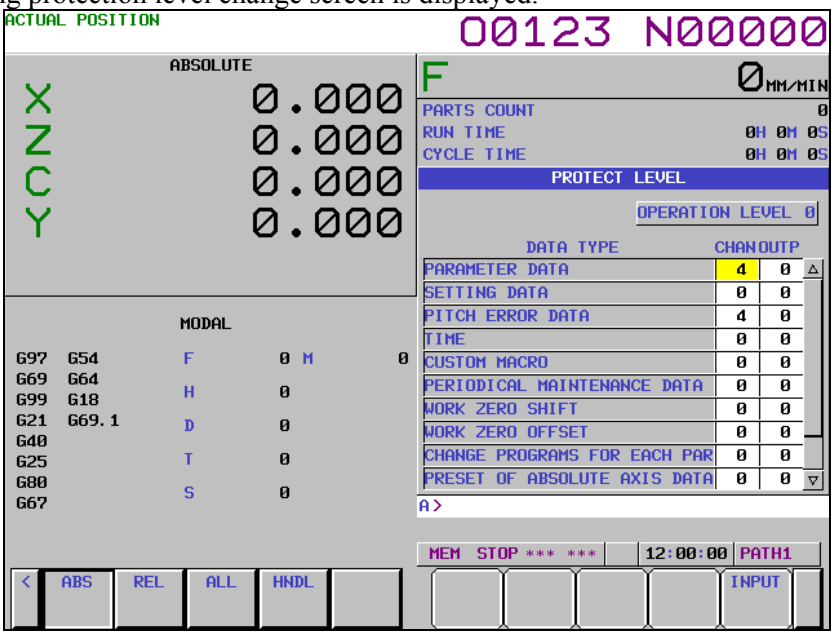


Fig. 12.3.11.3 (a) Protection level change screen (10.4-inch display unit)

5 Move the cursor to the change level or output level of a desired data item.

6 Key in a new desired level, then press soft key [INPUT].

NOTE
When the protection level of PMC data is set, soft key [SWITCH PMC] is used to switch between PMC paths to be set, for multi-path PMC.

Explanation

When the protection level of a data item is higher than the current operation level, the protection level of the data item cannot be changed.
The protection level of a data item cannot be changed to a protection level higher than the current operation level.
For each of the following types of data, you can set a data protection level. There are the following two types of data protection levels:

- Change protection level
  Sets the protection level used when data is changed.

- Output protection level
  Sets the protection level used when data is output to an external unit.

As a protection level, you can set a value of 0 (low) to 7 (high).

Table 12.3.11.3 (a) Protection level of each type of data

Type of data	Initial protection level	
	Change	Output
Custom macro variable data <CUSTOM MACRO> (including variable data dedicated to the macro executor)	0	0
Periodical maintenance data <PERIODICAL MAINTENANCE DATA>	0	0
Tool management data <TOOL LIFE TIME DATA>	0	0
Tool offset data <TOOL OFFSET DATA> (For each type when tool geometry compensation and tool wear compensation are treated differently)	0	0
Clock data <TIME>	0	0
Workpiece origin shift amount data <WORK ZERO SHIFT>	0	0
Workpiece origin offset data <WORK ZERO OFFSET>	0	0
Ethernet setting data <ETHER NET SETTING>	0	0
Fixture offset data <FIXTURE OFFSET DATA>	0	0
Parameter data <PARAMETER DATA>	4	0
Settings <SETTING DATA>	0	0
Pitch error compensation data <PITCH ERROR DATA> (including 3-dimensional error compensation data)	4	0
Parameter data for Power Mate CNC manager function <PARAMETER DATA FOR POWER MATE CNC MANAGER>	0	0
Each part program <PROGRAMS FOR EACH PARTS>	0	0
Part program editing operation <CHANGE PROGRAMS FOR EACH PARTS>	0	0
Absolute coordinate preset operation <PRESET OF ABSOLUTE AXIS DATA>	0	0

Table 12.3.11.3 (b) Protection level of PMC data

Type of data	Initial protection level	
	Change	Output
Composition parameter	0	0
Setting (online)	0	0
Setting (each path)	0	0
Sequence program	0	0
PMC parameter	0	0
Timer	0	0
Counter	0	0
Keep relay	0	0
Keep relay (system)	0	0
Data table	0	0
Data table control	0	0
PMC memory	0	0
I/O configuration	0	0
I/O Link group selection	0	0
Registration of I/O device	0	0

NOTE
1 For some types of data, the output function is not provided.
2 When the protection level of data is higher than the current operation level, the protection level cannot be changed.
3 The protection level of data cannot be changed to a level higher than the current operation level.

NOTE
4 Settable types of data increase or decrease, depending on the option configuration.
5 For details on the protection level of PMC data, refer to “PMC Programming Manual (B-64513EN)”.
6 Data related to tool information on the EACH TOOL DATA screen for the tool management function is protected by protecting tool offset data.
7 The type of tool offset data put in effect varies depending on the tool compensation value memory used.
8 To change the protection level for each part program, do so on the PROGRAM FOLDER screen rather than on the PROTECT LEVEL screen.
9 Part program editing includes program editing for the MDI mode.
10 Presetting absolute coordinates causes preset workpiece coordinate system values to be protected.
11 During tool offset data input/output, if any tool offset data type is not allowed to be changed or output to the outside, it is processed as follows:
- Input : Any data type other than those whose change is not allowed is changed.
- Output : Any data type other than those whose change is not allowed is output.

12.3.11.4 Setting the change protection level and output protection level of a program

The display/operations indicated below can be performed from the directory screen.
The change protection level and output protection level of each part program are displayed.
The change protection level and output protection level of each part program can be changed.

Setting the change protection level and output protection level of a program

Procedure

1 Press function key .
2 Press soft key [FOLDER].
The program folder screen shown below is displayed.

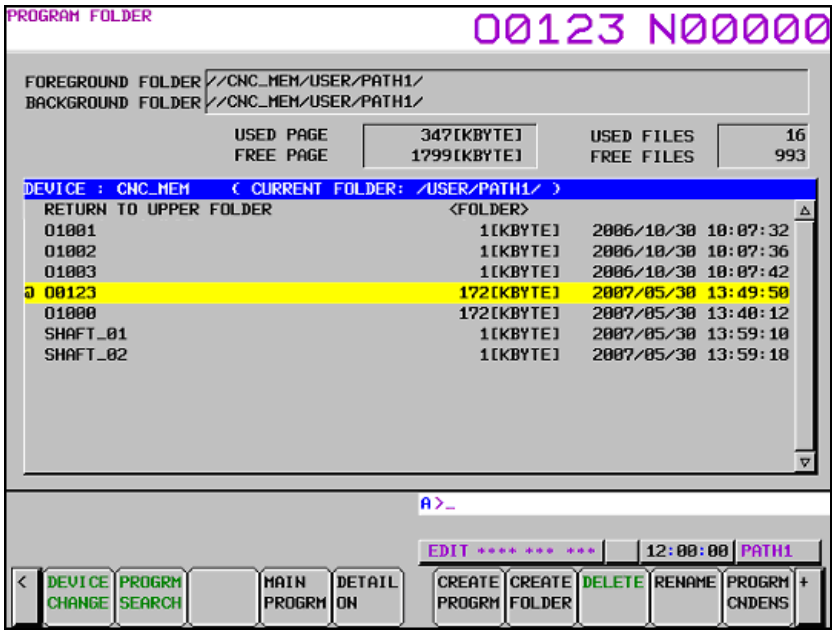


Fig. 12.3.11.4 (a) Program folder screen (10.4-inch display unit)

3 Press soft key [(OPRT)].
4 Press soft key [DETAIL ON]. The screen display switches to the detail display screen.
5 Move the cursor to a desired program.
6 Press the continuous menu key ▷ several times until [CHANGE ATTR] is displayed.
7 Press soft key [CHANGE ATTR].
8 Key in a new desired level, then press soft key [CHANGE LEVEL].
9 To change the output protection level, key in a new desired level, then press soft key [OUT LEVEL].

Explanation

The change protection level (0 to 7) and output protection level (0 to 7) are displayed as "CHANGE PROTECTION LEVEL VALUE/OUTPUT PROTECTION LEVEL".

NOTE
1 When the protection level of data is higher than the current operation level, the protection level cannot be changed.
2 The protection level of data cannot be changed to a level higher than the current operation level.
3 A protection level can be set only for those part programs that are held on the "CNC_MEM" device.

12.3.12 Precision Level Selection

An intermediate precision level between the parameters for emphasis on velocity (precision level 1) and the parameters for emphasis on precision (precision level 10) set on the machining parameter tuning screen (AI contour) can be selected. As shown in the Fig. 12.3.12 (a), the levels are proportionally linear, and an intermediate level can be selected so that optimal parameters can be automatically calculated to perform machining.

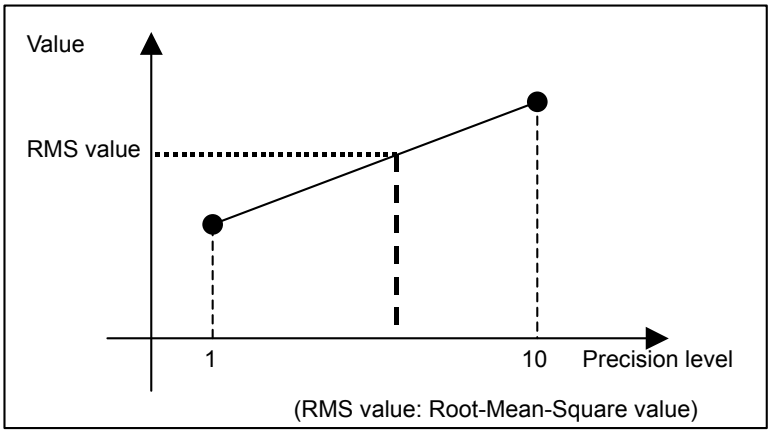


Fig. 12.3.12 (a) Image of "level"

Procedure for precision level selection

1 Select the MDI mode.

2 Press function key .

3 Press soft key [PRECI LEVEL].

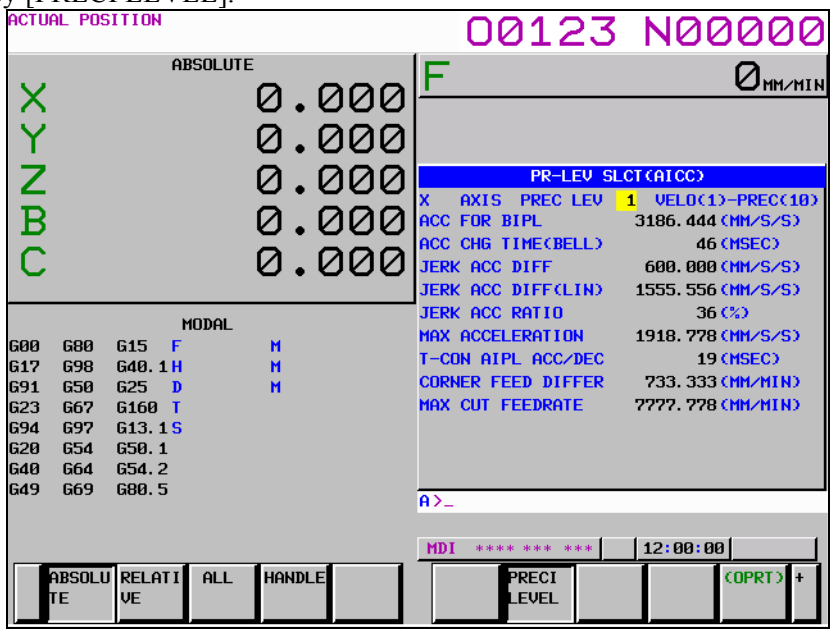


Fig. 12.3.12 (b) Precision level selection screen (10.4-inch display unit)

4 To change the precision level, key in a desired precision level (1 to 10), then press the  key on the MDI unit.

5 When the precision level is changed, a RMS value is obtained from the velocity-emphasized parameter set and precision-emphasized parameter set for parameter modification.
For the modified parameters, see the description of the machining parameter tuning.

6 If there is an axis in addition to the currently displayed axes, press page key  or  several times to display the screen for the axis.

12.3.13 Machining Level Selection

12.3.13.1 Smoothing level selection

An intermediate smoothing level between the parameters for smoothing level 1 and the parameters for smoothing level 10 set on the machining parameter tuning screen (smoothing) can be selected. As shown in the Fig. 12.3.13.1 (a), the levels are proportionally linear, and an intermediate level can be selected so that optimal parameters can be automatically calculated to perform machining.

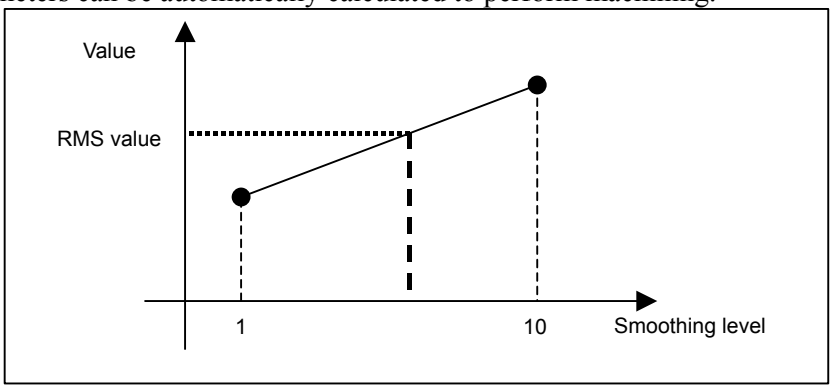


Fig. 12.3.13.1 (a) Image of "level"

Procedure for smoothing level selection

1 Select the MDI mode.
2 Press function key .
3 Press soft key [PRECI LEVEL].
4 Press soft key [SMOOTH LEVEL].

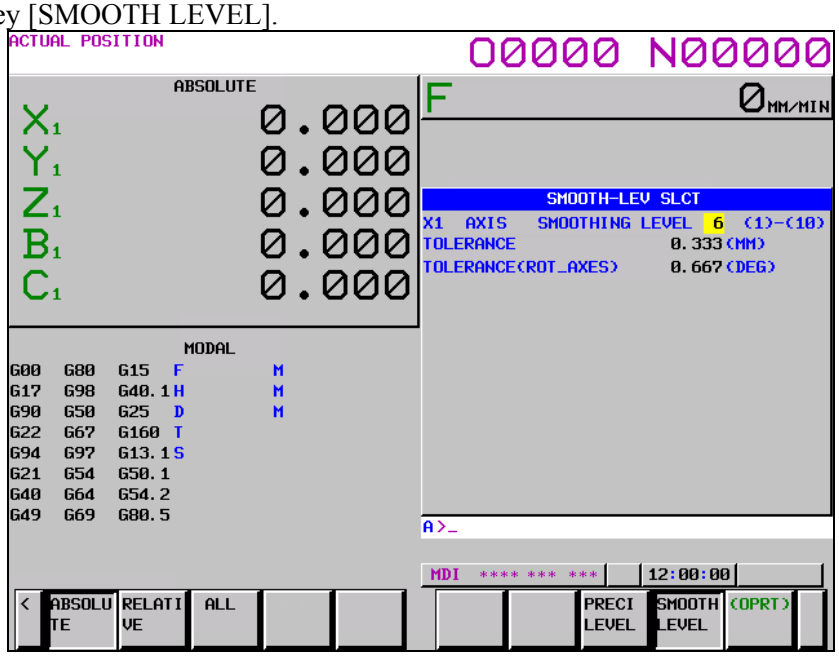


Fig. 12.3.13.1 (b) Smoothing level selection screen (10.4inch)

5 To change the smoothing level, key in a desired smoothing level (1 to 10), then press the  key on the MDI unit.
6 When the smoothing level is changed, a RMS value is obtained from the smoothing level 1 parameter set and smoothing level 10 parameter set for parameter modification.
For the modified parameters, see the description of the machining parameter tuning.

7 If there is an axis in addition to the currently displayed axes, press page key PAGE or PAGE several times to display the screen for the axis.

12.3.13.2 Precision level selection

For details of precision level selection, See Subsection, “Precision Level Selection”.

12.3.14 Machining Quality Level Selection

Machining quality level selection allows the precision level and smoothing level to be adjusted intuitively and easily.
To display the machining quality level selection screen, set bit 6 (QLS) of parameter No. 11350 to 1.

NOTE
The machining quality level selection screen cannot be displayed on the 8.4-inch display unit.
On these display units, only the machining level selection screen can be displayed.

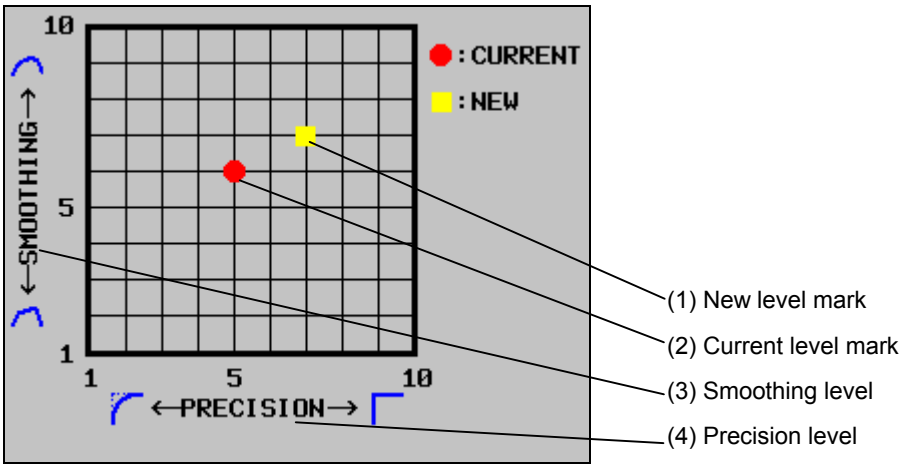


Fig. 12.3.14 (a) Quality level selection

(1) New level mark
Yellow square: Indicates the setting to be selected. (Cursor position)
(2) Current level mark
Red circle: Indicates the current setting.
(3) Smoothing level
Vertical axis: Indicates the smoothing level (1 to 10).
(4) Precision level
Horizontal axis: Indicates the precision level (1 to 10).

Procedure for machining quality level selection screen

1 Enable parameter writing.
2 Press function key OFS SET.
3 Press soft key [QUALITY SELECT].

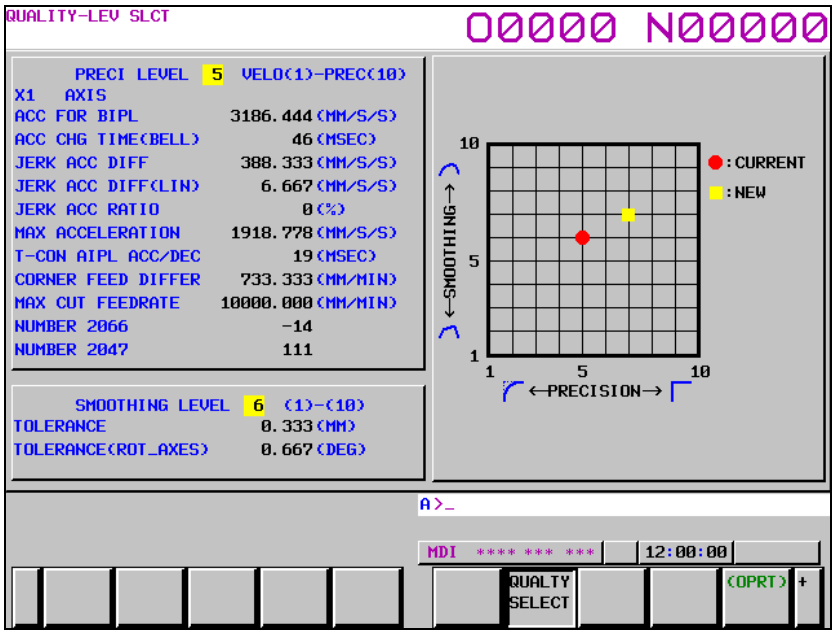


Fig. 12.3.14 (b) Machining quality level selection screen (10.4 inch)

4 Use cursor keys to move the new level mark and select the level.
(The new level mark moves.)

5 Press soft key [APPLY] or MDI key 

to set the level.

(The current level mark moves to the position of the new level mark.)
Whether to enable or disable MDI key operation can be switched by setting the relevant parameter.

6 The set precision level and smoothing level are reflected in each setting on the PRECI LEVEL and SMOOTHING LEVEL screens displayed on the left side of the screen.

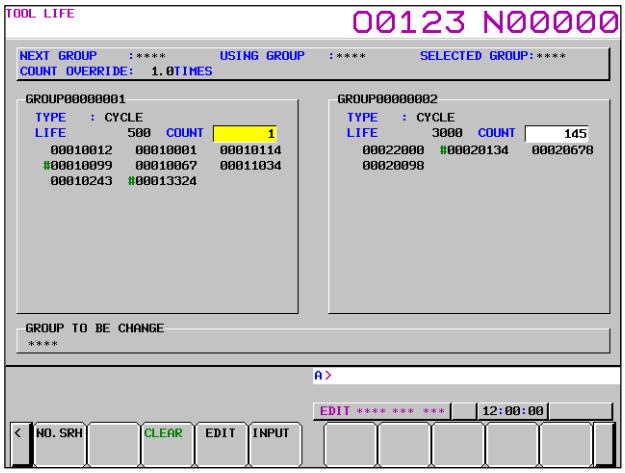
7 When the precision level or smoothing level is changed, an RMS value is obtained using the parameter settings for precision levels 1 and 10 and smoothing levels 1 and 10 and effective parameters are changed. For the changed parameters, see the description of the machining parameter tuning screen. When there is an axis other than the currently displayed axis, press a page key several times to display the screen for the desired axis.

12.3.15 Displaying and Setting Tool Life Management Data

Displaying tool life management data on a screen enables the current status of tool life management to be grasped. Also on the screen, tool life management data can be edited. The screen is either:

- Tool life management (list screen) or
- Tool life management (group editing screen)

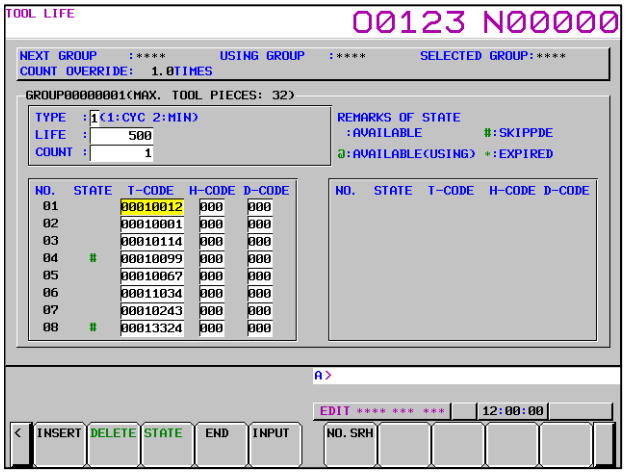
Overview



List screen

EDIT

END



Group editing screen

Tool life management (group editing screen)
Displayed items:
- NEXT GROUP - SELECTED GROUP
- MAX.TOOL PIECES - LIFE - COUNT
- T-CODE - H-CODE and D-CODE

Functions:
- Setting tool data - Setting a tool status
- Selecting too skip
- Setting a life count type, life value, and tool life counter

Tool life management (list screen)
Displayed items:
- NEXT GROUP
- SELECTED GROUP
- GROUP NO.
- LIFE
- TOOL MANAGEMENT STATUS
- GROUP TO BE CHANGE

Functions:
- Searching for groups
- Clearing execution data
- Setting the tool life counter

Pressing horizontal soft key [EDIT] or [END] switches from the list screen to the group editing screen or vice versa.

M

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1), the following parameters can be used to display arbitrary groups and remaining set values.

- If bit 5 (TGN) of parameter No. 6802 = 1
  An arbitrary group is displayed on the list and group editing screens.
- If bit 3 (GRP) of parameter No. 6802 = 1
  Remaining set values are displayed on the group editing screen. The group editing screen always displays H and D codes.

T

The T series is provided with two tool change types, turret type and ATC type. For the T series, displays vary depending on which tool change type is in use.
If the turret type is in use (bit 3 (TCT) of parameter No. 5040 = 0)

- Neither the H code nor the D code is used; neither is displayed.
- No arbitrary group can be used; no arbitrary group is displayed.

If the ATC type is in use (bit 3 (TCT) of parameter No. 5040 = 1)

- The D code is displayed on the group editing screen.
- If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1) and bit 5 (TGN) of parameter No. 6802 = 1, arbitrary groups are displayed on the list and group editing screens.

If the tool life management B function is enabled (bit 4 (LFB) of parameter No.6805 = 1) and bit 3 (GRP) or parameter No. 6802 = 1, remaining set values are displayed on the group editing screen.

12.3.15.1 Tool life management (list screen)

This screen can display the life management status of all tools in tool groups and whether the life of the tool groups has expired. It also enables you to set tool life counters and clear execution data.

Displays on the list screen

Procedure

1 Press function key .

2 Press the continuous menu key  several times to display soft key [TOOL LIFE].

3 Press soft key [TOOL LIFE].

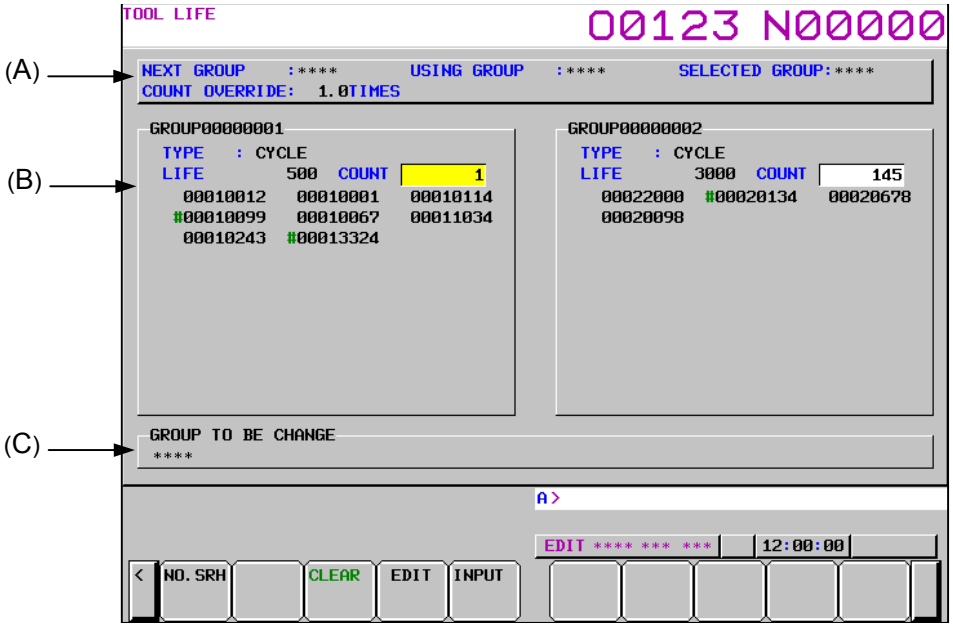


Fig. 12.3.15.1 (a) Displaying tool life management (list screen) (10.4-inch display unit)

- Contents of (A)

(A) displays tool group numbers and an override value. If there is no tool group to display, ****, instead of tool group numbers, is displayed.

NEXT GROUP : Tool group number for which life counting is started by the next M06 command.

USING GROUP : Tool group number for which life counting is currently under way.

SELECTED GROUP : Tool group number for which life counting is currently under way or life counting has been performed most recently.

COUNT OVERRIDE : "1.0TIMES" is displayed if the tool life counter override signal is disabled (bit 2 (LFV) of parameter No. 6801 = 0).

NOTE
If arbitrary group numbers are enabled, NEXT GROUP, USING GROUP, and SELECTED GROUP are each represented with an arbitrary group number rather than the tool group number.

- Contents of (B)

(B) displays the set life value, the current content of the tool life counter, and the registered tool numbers (in the order they are used) for each tool group. If the life count type is specification by duration, the measurement unit used in displaying and specifying set life values and tool life counter values is selected according to the setting of bit 0 (FCO) of parameter No. 6805 as listed below.

Bit 0 (FCO) of parameter No. 6805	0	1
Measurement unit used in displaying and specifying set life values and tool life counter values	1 minute	0.1 minutes

The following table lists the prefixes used with tool numbers.

Tool status	Tool in use	Tool not in use
Life remaining	@	No indication
Skip	#	#
Life expired	*	*

NOTE
1 The tool life counter indicates the count value for the tool indicated with @.
2 If bit 3 (EMD) of parameter No. 6801 = 0, a tool number remains prefixed with @ even if the life of the tool has expired until another tool is selected.
3 If bit 3 (EMD) of parameter No. 6801 =1, the following differences can occur depending on the type of the life counter in use.
- If the life counter type is specification by duration, the prefix is changed to "*" (life expired) as soon as the tool life expires.
- If the life counter type is specification by count, the counter is incremented by one at the end of a program (such as M02 or M30). So, the prefix is not changed to "*" (life expired) even if the tool life counter matches the life value. The symbol "*" (life expired) appears when the tool life count is incremented after the CNC is reset.

4 If bit 2 (ETE) of parameter No. 6804 = 1, the symbol "*" indicating the expiration of the life of the last tool appears on the tool life management screen when the life counter for the last tool of the tool group of interest matches the life value. This way, the information about the last tool in the FOCAS2 or PMC window indicates that the life of the tool has expired if the tool change signal TLCH <Fn064.0> is 1.

If arbitrary group numbers are enabled, an arbitrary group number is displayed in the parentheses beside the tool group number. If no arbitrary group number is specified, "********" is displayed instead.

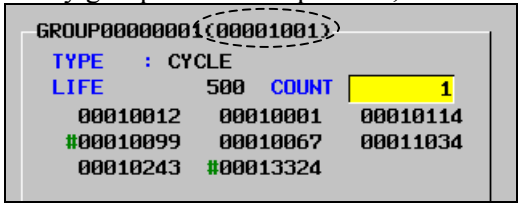


Fig. 12.3.15.1 (b) Displaying arbitrary group numbers

Arbitrary group numbers are enabled by setting the following parameters.

M

- The tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1).
- Arbitrary group numbers are enabled (bit 5 (TGN) of parameter No. 6802 = 1).

T

- The current tool change type is ATC (bit 3 (TCT) of parameter No. 5040 = 1).
- The tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1).
- Arbitrary group numbers are enabled (bit 5 (TGN) of parameter No. 6802 = 1).

- Contents of (C)

(C) displays tool group numbers for which a tool change signal has been issued.
If there are so many tool group numbers that all the numbers cannot be displayed, some are omitted, and "**>>**" is displayed instead.
If there is no tool group number in need of change, "********" is displayed.

NOTE
If arbitrary group numbers are enabled, a tool group number in need of change is represented with an arbitrary group number rather than the tool group number.

Setting data on the list screen

Tool life management data can be specified in the reset state (both the OP and RST signals are "0"). However, setting bit 1 (TCI) of parameter No. 6804 to 1 enables tool life management data to be specified even during automatic operation (the OP signal is "1").

NOTE
As for USING GROUP or NEXT GROUP settings:
1) During automatic operation (OP signal = "1" and bit 1 (TCI) of parameter No. 6804 = 1), only the tool life counter can be changed.
2) In the reset state (OP signal = "0" and RST signal "0"), the following editing operation stops life management because it cannot be continued.
- Clearing execution data

Procedure

- Setting the tool life counter

The tool life counter can be set with a value, using the following methods.
Method 1
1 Place the cursor on the tool life counter for a desired tool group.
2 Enter the value from the keypad.
3 Press soft key [INPUT].

Method 2
1 Place the cursor on the tool life counter for a desired tool group.
2 Enter the value from the keypad.
3 Press 

key.

- Clearing execution data

All existing execution data for a tool group selected by the cursor can be cleared as follows:
1 Place the cursor on the tool group whose execution data you want to clear.
2 Press soft key [ERASE].
3 Press soft key [EXEC].

NOTE
Setting bit 4 (GRS) of parameter No. 6800 to 1 enables execution data for all registered tool groups to be cleared.

- Selecting tool groups

Tool groups can be selected using the following methods.
Method 1

1 Enter a tool group number from the keypad.
2 Press soft key [NO.SRH].

NOTE
If arbitrary group numbers are enabled, a tool group is selected by searching for an arbitrary group number rather than the tool group number.

Method 2

1 Press page key  or  to display desired groups.
2 Press cursor key  or  to move the cursor to the desired group at either the left or right.

- Switching to the group editing screen

Switch to tool life management (group editing screen).

1 Move the cursor to the tool group you want to edit:
2 Press soft key [EDIT].

12.3.15.2 Tool life management (group editing screen)

On this screen, it is possible to edit tool life management data (such as tool life value, tool life counter, and tool data) for the tool group of interest.

Displaying the group editing screen

Procedure

1 Place the cursor on the tool group you want to edit from the list screen.
2 Press soft key [(OPRT)].
3 Press soft key [EDIT].

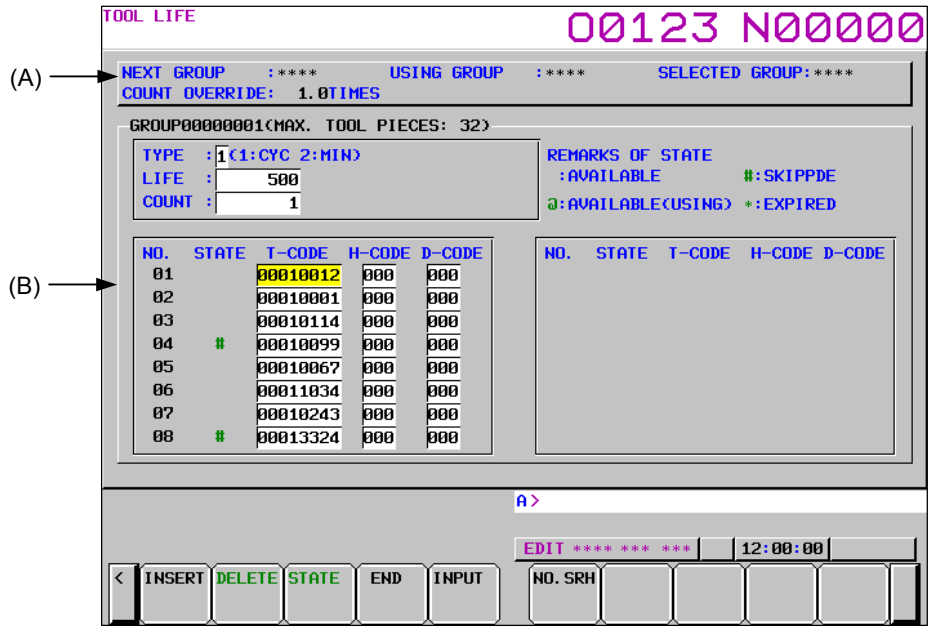


Fig. 12.3.15.2 (a) Displaying tool life management (group editing screen) (10.4-inch display unit)

NOTE
If no tool is registered with a tool group, none of a life count type, a life value, and a tool life counter value is displayed for the tool group.

- Contents of (A)

Like the counterpart of the list screen, (A) of the editing screen displays the next tool group number and override value. If there is no appropriate tool group, **** is displayed instead.

NEXT GROUP : Tool group for which life counting is started by the next M06 command.
USING GROUP : Tool group number for which life counting is currently under way.
SELECTED GROUP : Tool group number for which life counting is currently under way or life counting has been performed most recently.
COUNT OVERRIDE : "1.0TIMES" is displayed if the tool life counter override signal is disabled (bit 2 (LFV) of parameter No. 6801 = 0).

NOTE
If arbitrary group numbers are enabled, NEXT GROUP, USING GROUP, and SELECTED GROUP are each represented with an arbitrary group number rather than the tool group number.

- Contents of (B)

(B) details tool life management data related to a selected tool group as follows:

TYPE :

1	Specification by count
2	Specification by duration

LIFE : Tool life value
COUNT : Tool life counter
STATE :

Tool status	Tool in use	Tool not in use
Life remaining	@	No indication
Skip	#	#
Life expired	*	*

T-CODE : Tool number

M

H-CODE : Tool length compensation specification code
D-CODE : Cutter compensation specification code

T

H-CODE : No display.
D-CODE : Tool offset value specification code
if ATC type is in use (bit 3 (TCT) of parameter No. 5040 = 1)

NOTE

1 The tool life counter indicates the count value for the tool indicated with @.
2 If bit 3 (EMD) of parameter No. 6801 = 0, a tool number remains prefixed with @ even if the life of the tool has expired until another tool is selected.
3 If bit 3 (EMD) of parameter No. 6801 = 1, the following differences can occur depending on the type of the life counter in use.
- If the life counter type is specification by duration, the prefix is changed to "*" (life expired) as soon as the tool life expires.
- If the life counter type is specification by count, the counter is incremented by one at the end of a program (such as M02 or M30). So, the prefix is not changed to "*" (life expired) even if the tool life counter matches the life value. The symbol "*" (life expired) appears when the tool life count is incremented after the CNC is reset.

4 If bit 2 (ETE) of parameter No. 6804 = 1, the symbol "*" indicating the expiration of the life of the last tool appears on the tool life management screen when the life counter for the last tool of the tool group of interest matches the life value. This way, the information about the last tool in the FOCAS2 or PMC window indicates that the life of the tool has expired if the tool change signal TLCH <Fn064.0> is 1.

Displaying arbitrary groups and remaining set values

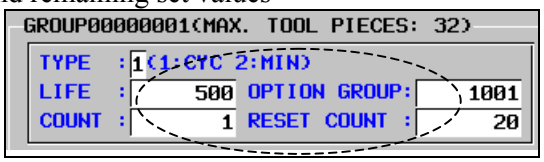


Fig. 12.3.15.2 (b) Displaying arbitrary group numbers and remaining set values on the editing screen

M

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1) and the following parameters are enabled, arbitrary groups and remaining set values are displayed.
OPTION GROUP : Arbitrary group number (if bit 5 (TGN) of parameter No. 6802 = 1)
REST COUNT : Remaining set value used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1)

T

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1)
Turret type (bit 3 (TCT) of parameter No. 5040 = 0)
OPTION GROUP : No display.
REST COUNT : Remaining set value used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1)

ATC type (bit 3 (TCT) of parameter No. 5040 = 1)
OPTION GROUP : Arbitrary group number (if bit 5 (TGN) of parameter No. 6802 = 1)
REST COUNT : Remaining set value used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1)

Setting data on the group editing screen

Tool life management data can be specified in the reset state (both the OP and RST signals are "0"). However, setting bit 1 (TCI) of parameter No. 6804 to 1 enables tool life management data to be specified even during automatic operation (the OP signal is "1").
The available editing operations are listed below.

M

Items that can be edited	Mode
Setting a life count type, life value, tool life counter, and tool data (T code, H code, and D code)	All modes
Adding tool numbers (T code)	MDI
Deleting all tool group data at a time	MDI
Deleting tool data (status, T code, H code, and D code)	MDI
Selecting tool skip	MDI
Specifying to clear tool data (life re-set)	MDI

T

Items that can be edited	Mode
Setting a life count type, life value, tool life counter, and tool data (T code and D code) However, the D code can be edited only if the ATC type is in use (bit 3 (TCT) of parameter No. 5040 = 1).	All modes
Adding tool numbers (T code)	MDI
Deleting all tool group data at a time	MDI
Deleting tool data (status, T code, and D code)	MDI
Selecting tool skip	MDI
Specifying to clear tool data (life re-set)	MDI

M
T

If no tool is registered with a tool group, none of a life count type, a tool life value, and a tool life counter value can be set for the tool group. First of all, add a tool number (T code).

NOTE
1 As for USING GROUP or NEXT GROUP editing:
<1> During automatic operation (OP signal = "1" and bit 1 (TCI) of parameter No. 6804 = 1), only the tool life counter can be changed.
<2> In the reset state (OP signal = "0" and RST signal "0"), the following editing operation stops life management because it cannot be continued.
- Adding tool numbers (T code)
- Deleting all tool group data at a time
- Deleting tool data (status, T code, H code, and D code)

2 The following editing operations may set the tool change signal to "1".
- Selecting tool skip for the last tool.
- Deleting tool numbers, resulting in any tool other than those whose life has expired or who have been skipped being not found in the tool group of interest.

NOTE
3 The following editing operations may reset the tool change signal to “0”.
- Adding tool numbers, leading to tools whose life has not expired being set in the tool group of interest.
- Selecting tool clear.

Procedure

- Setting a life count type, tool life value, tool life counter, tool data, arbitrary group number, and remaining set value

Setting a life count type, tool life value, tool life counter, tool data, arbitrary group number, and remaining set value

Method 1

1 Place the cursor on the desired item.
2 Enter a value from the keypad.
3 Press soft key [INPUT].

Method 2

1 Place the cursor on the desired item.
2 Enter a value from the keypad.
3 Press 

.

NOTE
1 Changing a tool life value or tool life counter does not affect the tool status or tool change signal.
2 Changing a life count type causes the tool life value and too life counter to be reset to 0.

M

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1) and the following parameters are enabled, arbitrary groups and remaining set values can be specified
OPTION GROUP : Arbitrary group number (if bit 5 (TGN) of parameter No. 6802 = 1)
REST COUNT : Remaining set value used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1)

T

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1)
Turret type (bit 3 (TCT) of parameter No. 5040 = 0)
OPTION GROUP : Cannot be set.
REST COUNT : It is possible to set a remaining set value to be used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1).
ATC type (bit 3 (TCT) of parameter No. 5040 = 1)
OPTION GROUP : Arbitrary group numbers can be set (if bit 5 (TGN) of parameter No. 6802 = 1).
REST COUNT : It is possible to set a remaining set value to be used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1).

- Adding tool numbers

Tool numbers can be added to a tool group as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) just before a tool number to be added.
3 Enter the tool number from the key pad.
4 Press soft key [INSERT].

(Example)

Adding tool number 1550 between numbers 1 and 2 (for the M series)

1 Move the cursor to the data for number 1, enter “1550”, and press [INSERT].

NO.	STATE	T-CODE	H-CODE	D-CODE
01		00010012	000	000
02		00010001	000	000
03		00010114	000	000

2 The entered T code 1550 is inserted in the position of number 2. The H and D codes are reset to 0.

NO.	STATE	T-CODE	H-CODE	D-CODE
01		00010012	000	000
02		00001550	000	000
03		00010001	000	000

- Deleting all tool group data at a time

All tool group data can be deleted at a time as follows:

1 Select the MDI mode.
2 Select the tool group from which you want to delete all tool data at a time.
3 Press soft key [DELETE].
4 Press soft key [GROUP].
5 Press soft key [EXEC].

- Deleting tool data

Tool data can be deleted from a tool group as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) you want to delete.
3 Press soft key [DELETE].
4 Press soft key [<CURS>].

NOTE
1 Deleting all tools from a tool group is equivalent to deleting the tool group itself.
2 Deleting a tool indicated with @ (in use) results in @ being moved to the previous tool whose life has expired or which has been skipped.

- Selecting tool skip

Tool data can be placed in a skip state as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) for a tool you want to skip.
3 Press soft key [STATE].
4 Press soft key [SKIP].

- Specifying to clear tool data (life re-set)

Tool data state can be cleared as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) for a tool you want to clear.
3 Press soft key [STATE].
4 Press soft key [CLEAR].

- Selecting a tool group

A tool group can be selected as follows:

Method 1

1 Enter a tool group number from the keypad.

2 Press soft key [NO.SRH].

NOTE
If arbitrary group numbers are enabled, a tool group is selected by searching for an arbitrary group number rather than the tool group number.

Method 2

1 Press page key  or  to display the target tool group.

- Switching to the list screen

The tool life management (list screen) can be resumed as follows:

1 Press soft key [END].

12.3.16 Displaying and Setting Workpiece Setting Error Compensation Data

An amount of error used in workpiece setting error compensation can be set on the workpiece setting error screen.
The workpiece setting error screen is displayed as follows:

1 Press function key .

2 Press chapter selection soft key [WORD SET ER].
The workpiece setting error screen appears.

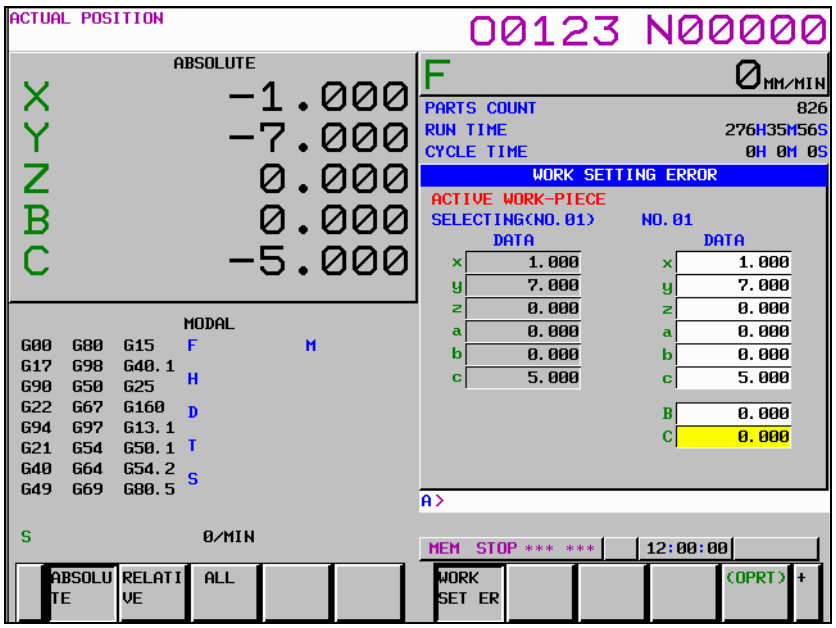


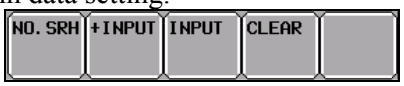
Fig. 12.3.16 (a) Workpiece setting error screen (10.4-inch display unit)

Up to 7 sets (No. 01 to No. 07) of different amounts of workpiece setting error can be specified. In addition, it is possible to specify an amount of error (No. 00 (COMMON)) to be added to an error in the 7 sets in common.
The X-direction error Δx, Y-direction error Δy, and Z-direction error Δz are specified, respectively, in the x, y, and z fields. The rotational errors Δa, Δb, and Δc are specified, respectively, in the a, b, and c fields.
Table rotation axis positions 1 and 2 are specified in the fields under the a, b, and c fields (C and B in the figure).
Rotation axis names specified in parameters Nos. 19681 and 19686 are displayed in the axis name fields.

Table rotation axis positions 1 and 2 are specified for rotation axes. No position must be specified for axes (including virtual axes) other than table rotation axes. So, no such item is displayed for setting.
Table rotation axis position setting items are displayed under the following conditions.

- It must have been correctly specified which axis is a table rotation axis.
  If parameter No. 19680 = 12 (table rotation type), axis numbers must have been specified correctly in parameters Nos. 19681 and 19686.
  If parameter No. 19680 = 21 (mixed type), an axis number must have been specified correctly in parameter No. 19686.
- Bit 0 of parameter No. 1006 must be 1 (rotation axis) for the axes specified above.

The following soft key can be used in data setting.



- After a workpiece setting error number you want to display is entered, pressing [NO.SRH] causes a setting screen for the target workpiece setting error to appear.
- After a number is entered, pressing soft key [+INPUT] causes the number to be added.
- After a number is entered, pressing soft key [INPUT] causes the number to be set.
- Pressing soft key [ERASE] causes the next soft key to appear.



- After a workpiece setting error number you want to delete is entered, pressing soft key [NO.] causes the data corresponding to the error number to be deleted.
- Pressing soft key [ALL] causes the data corresponding to all workpiece setting error numbers to be deleted.

12.3.17 Displaying and Setting Pattern Data Inputs

Described below are a method for displaying machining menus (pattern menus) created by machine tool builders and a method for setting them. The descriptions are based on an example. For actual pattern menus and pattern data, refer to manuals released by respective machine tool builders.

Displaying pattern data and pattern menus

Given below is a procedure for displaying pattern menus.

1 Press function key .
2 Press the continuous menu key .
3 Press soft key [PATTERN MENU].
  The pattern menu screen shown below appears.

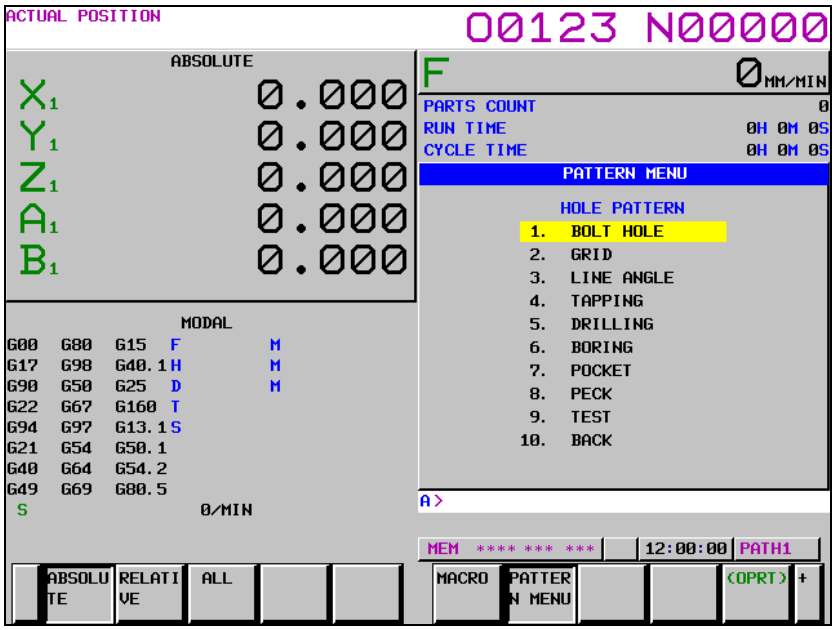


Fig. 12.3.17 (a) Pattern menu screen (10.4-inch display unit)

On this screen, a pattern to be used can be selected.
The following two methods can be used to select patterns.

- Using the cursor

  1 Move the cursor to a pattern name you want to select, using cursor key ↑ or ↓, and then press soft key [SELECT] or INPUT.

- Specifying a pattern number

  1 Enter a number displayed at the left side of a pattern name, and press soft key [SELECT] or INPUT.

  The custom macro screen (pattern data screen) shown below appears.

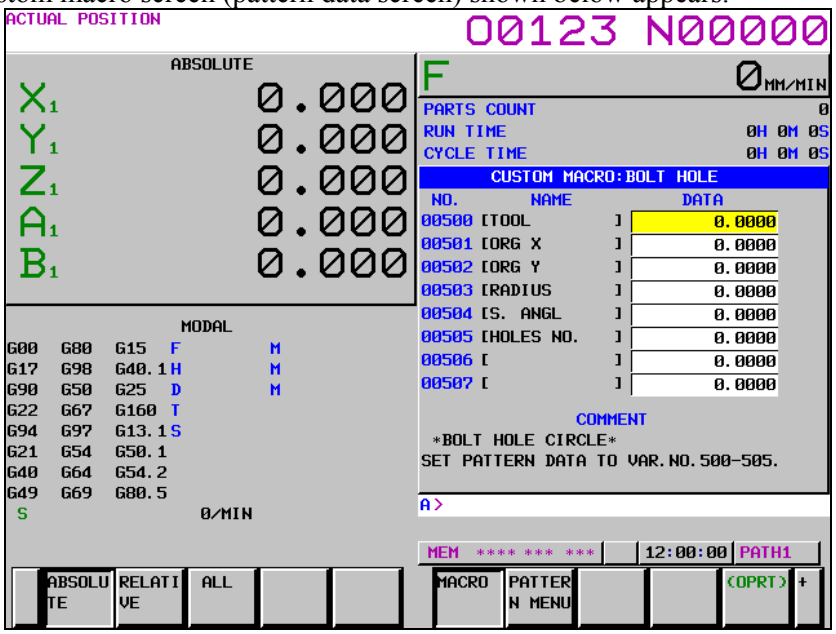


Fig. 12.3.17 (b) Custom macro screen (pattern data) (10.4-inch display unit)

2 Enter the necessary pattern data, and press INPUT.

3 After entering all necessary data, select the MEMORY mode and press the cycle start button. Machining begins.

Explanation

- **Explanations about the pattern menu screen**

HOLE PATTERN

An arbitrary character string consisting of 12 or less characters can be displayed as a menu title.

- BOLT HOLE

An arbitrary character string consisting of 10 or less characters can be displayed as a pattern name.

Machine tool builders should program menu title and pattern name character strings, using custom macros, and save them in program memory.

- **Explanations about the custom macro screen (pattern data screen)**

BOLT HOLE

An arbitrary character string consisting of 12 or less characters can be displayed as a pattern data title.

- TOOL

An arbitrary character string consisting of 10 or less characters can be displayed as a variable name.

- *BOLT HOLE CIRCLE*

The comment text consists of nine 12-character blocks, or the comment can be up to 12 lines (10.4-inch display unit) or 8 lines (8.4-inch display unit) with one block counted as one line.

Machine tool builders should program variable name and comment text character strings, using custom macros, and save them in program memory.

12.3.18 Setting and Displaying Data When the Tool Offset for Milling and Turning Function Is Enabled

The tool offset for milling and turning function enables offset data to be displayed and operated on a tool offset screen common to milling and turning systems.

Setting and displaying tool offset values

Procedure

1 Press function key .

In multi-path control, select the path for which to set tool offset values, using the path selection switch in advance.

2 Press chapter selection soft key [OFFSET].

Alternatively, press function key  several times until the tool offset screen appears.

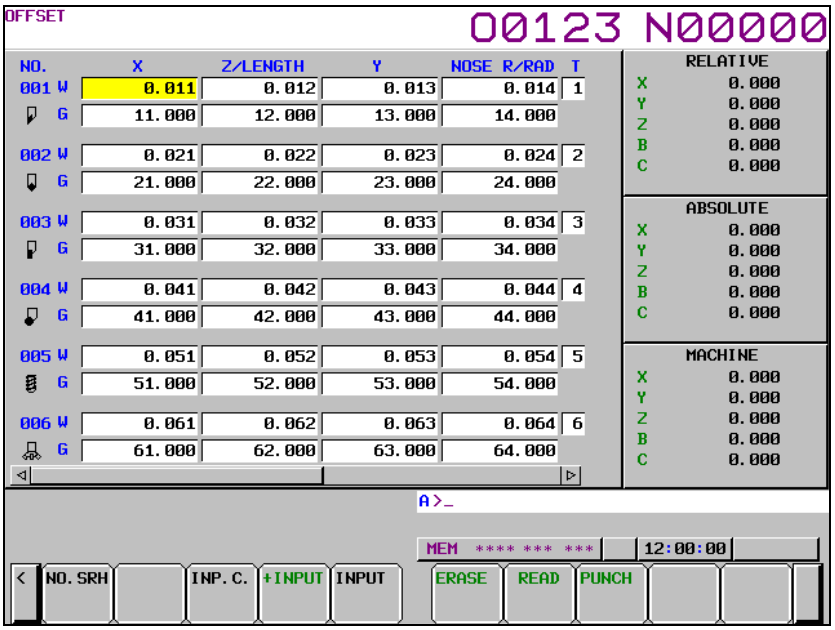


Fig. 12.3.18 (a) Tool offset screen (10.4-inch display unit)

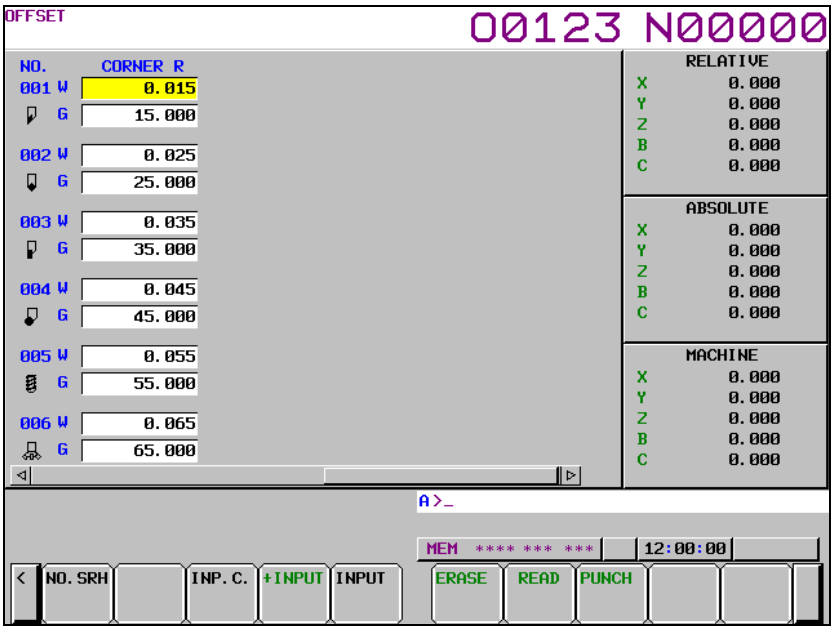


Fig. 12.3.18 (b) Tool offset screen (10.4-inch display unit)

X : Set the X-axis tool offset value.
Z/LENGTH : Set the Z-axis tool offset value or tool length compensation value.
Y : Set the Y-axis tool offset value.
NOSE R/RAD : Set the tool nose radius compensation value or tool radius compensation value.
T : Set the virtual tool nose direction.
CORNER R : Sets the corner-R offset value.
(This value can be set when the cutting point command option is enabled.)

3 Press page and cursor keys to move the cursor to the offset value you want to set or change. Alternatively, enter the number of the offset value you want to set or change and press soft key [NO.SRH].

4 To set the offset value, enter the value and press soft key [INPUT].
To change the offset value, enter a value to be incremented to or decremented from the current setting and press soft key [+INPUT].
Alternatively, enter a new offset value and press soft key [INPUT].

Explanation

- Decimal point programming

The offset value can be entered with a decimal point.

- Another setting method

The tool offset value can be input/output also by using external I/O devices.
See Chapter 8, “DATA I/O”, in Part III.
The tool offset value can also be set by using the direct input of offset value measured B, tool offset value counter input, and other functions.

- Number of tool offsets

Sixteen tool offsets are available as standard.
The number of tool offsets can be optionally expanded to 32, 64, or 999.
In a multi-path system, one of the above numbers of tool offsets can be selected for each path.

- Disabling offset value input

By setting bits 0 (WOF) and 1 (GOF) of parameter No. 3290, the input of tool wear compensation values and tool geometry compensation values is disabled.
The input of the tool offset values in any specified range from the MDI can be disabled by setting the number of the first target tool offset value in parameter No. 3294 and the number of target tool offset values starting from the first value in parameter No. 3295.
If offset values are entered continuously, they are handled as follows:

1) If an offset value that can be input is input, then an offset value that cannot be input is input, a warning is output, but the offset value in the input enable range is set.
2) If an offset value that cannot be input is input, then an offset value that can be input is input, a warning is output, and the offset values are not set.

- NOSE R/RAD, T display

When the tool radius/tool nose radius compensation option is not set, NOSE R/RAD, T are not displayed.

Screens of a 15/19-inch display unit

12.3.19 Displaying and Entering Setting Data (15/19-inch Display Unit)

Data such as the TV check flag and punch code is set on the setting data screen. On this screen, the operator can also enable/disable parameter writing, enable/disable the automatic insertion of sequence numbers in program editing, and perform settings for the sequence number comparison and stop function.
See Section, “AUTOMATIC INSERTION OF SEQUENCE NUMBERS” for automatic insertion of sequence numbers.
See Subsection, “Sequence Number Comparison and Stop” for the sequence number comparison and stop function. This subsection describes how to set data.

Procedure for setting the setting data

Procedure

1 Select the MDI mode.

2 Press function key 

.

3 Press vertical soft key [SETTING] to display the setting data screen.
This screen consists of several pages.
Press page key 

or 

until the desired screen is displayed.
An example of the setting data screen is shown below.

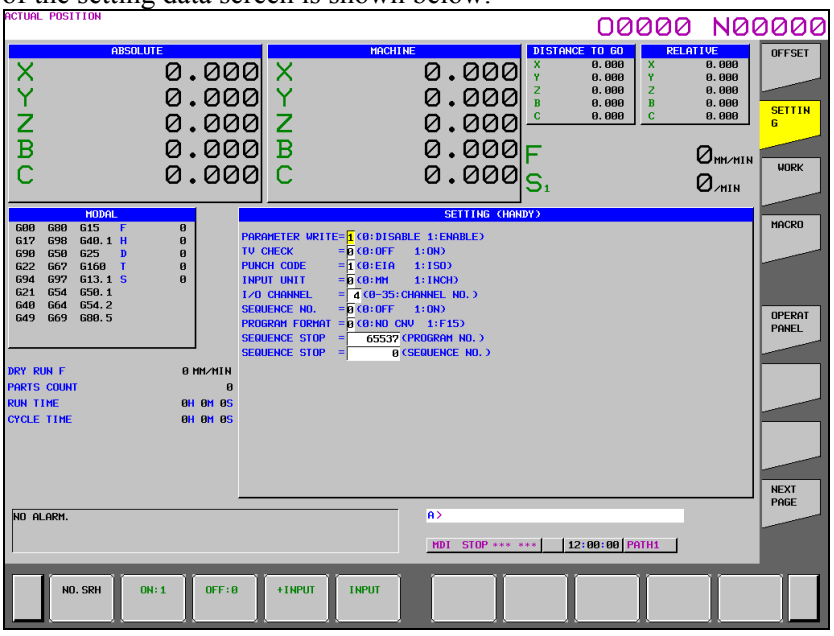


Fig. 12.3.19 (a) SETTING (HANDY) screen (15-inch display unit)

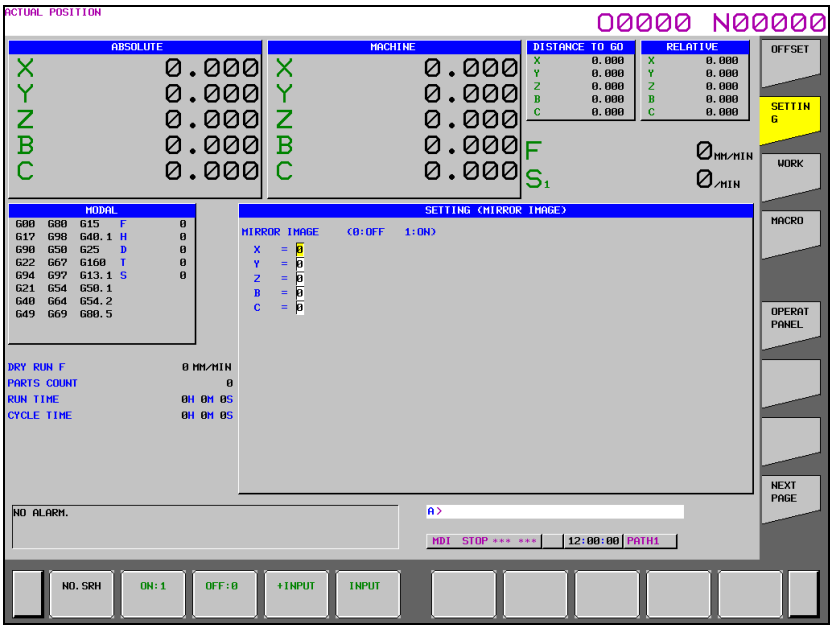
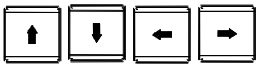


Fig. 12.3.19 (b) SETTING (MIRROR IMAGE) screen (15-inch display unit)

4 Move the cursor to the item to be changed by pressing cursor keys  .

5 Enter a new value and press horizontal soft key [INPUT].

Explanation

- PARAMETER WRITE

Setting whether parameter writing is enabled or disabled.
0 : Disabled
1 : Enabled

- TV CHECK

Setting to perform TV check.
0 : No TV check
1 : Perform TV check

- PUNCH CODE

Setting code when data is output through reader/puncher interface.
0 : EIA code output
1 : ISO code output

- INPUT UNIT

Setting a program input unit, inch or metric system
0 : Metric
1 : Inch

- I/O CHANNEL

Using channel of reader/puncher interface.
0 : Channel 0
1 : Channel 1
2 : Channel 2

- SEQUENCE NO.

Setting of whether to perform automatic insertion of the sequence number or not at program edit in the EDIT mode.
0 : Does not perform automatic sequence number insertion.
1 : Perform automatic sequence number insertion.

- PROGRAM FORMAT

Setting of whether to use the Series 15 format.
0: Uses the standard format.
1: Uses the Series 15 format.
For the Series 15 format, refer to Chapter, “MEMORY OPERATION USING Series 15 FORMAT” in the OPERATOR’S MANUAL (T series) or Chapter, “MEMORY OPERATION USING Series 15 FORMAT” in the OPERATOR’S MANUAL (M series).

- SEQUENCE STOP

Setting the sequence number with which the operation stops for the sequence number comparison and stop function and the number of the program to which the sequence number belongs.

- MIRROR IMAGE

Setting of mirror image ON/OFF for each axes.
0 : Mirror image off
1 : Mirror image on

- Others

Page key  or  can also be pressed to display the SETTING (TIMER) screen. See Subsection, “Displaying and Setting Run Time, Parts Count, and Time (15/19-inch Display Unit)” for this screen.

12.3.20 Sequence Number Comparison and Stop (15/19-inch Display Unit)

If a block containing a specified sequence number appears in the program being executed, operation enters single block mode after the block is executed.

Procedure for sequence number comparison and stop

Procedure

1 Select the MDI mode.
2 Press function key .
3 Press vertical soft key [SETTING].
4 Press page key  or  several times until the screen in the Fig. 12.3.20 (a) is displayed.

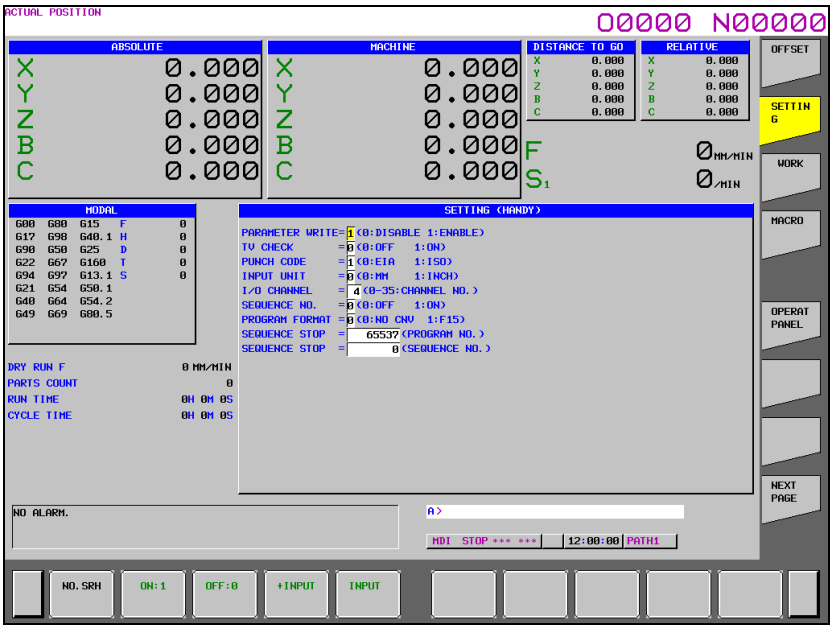


Fig. 12.3.20 (a) SETTING (HANDY) screen (15-inch display unit)

5 Enter in (PROGRAM NO.) for SEQUENCE STOP the number (1 to 99999999) of the program containing the sequence number with which operation stops.

6 Enter in (SEQUENCE NO.) for SEQUENCE STOP (with eight or less digits) the sequence number with which operation is stopped.

7 When automatic operation is executed, operation enters single block mode at the block containing the sequence number which has been set.

Explanation

- Sequence number after the program is executed

After the specified sequence number is found during the execution of the program, the sequence number set for sequence number compensation and stop is decremented by one.

- Exceptional blocks

If the predetermined sequence number is found in a block in which all commands are those to be processed within the CNC control unit, the execution does not stop at that block.

```
[Example]   N1 #1=1 ;
            N2 IF[#1 EQ 1]GOTO 08 ;
            N3 GOTO 09 ;
            N4 M98 P1000 ;
            N5 M99 ;
```

In the example shown above, if the predetermined sequence number is found, the execution of the program does not stop.

- Stop in the canned cycle

If the predetermined sequence number is found in a block which has a canned cycle command, the execution of the program stops after the return operation is completed.

- When the same sequence number is found several times in the program

If the predetermined sequence number appears twice or more in a program, the execution of the program stops after the block in which the predetermined sequence number is found for the first time is executed.

- Block to be repeated a specified number of times

If the predetermined sequence number is found in a block which is to be executed repeatedly, the execution of the program stops after the block is executed specified times.

12.3.21 Displaying and Setting Run Time, Parts Count, and Time (15/19-inch Display Unit)

Various run times, the total number of machined parts, number of parts required, and number of machined parts can be displayed.
This data can be set by parameters or on this screen (except for the total number of machined parts and the time during which the power is on, which can be set only by parameters).
This screen can also display the clock time. The time can be set on the screen.

Procedure for Displaying and Setting Run Time, Parts Count and Time

Procedure

1 Select the MDI mode.
2 Press function key .
3 Press vertical soft key [SETTING].
4 Press page key  or  several times until the screen in the Fig. 12.3.21 (a) is displayed.

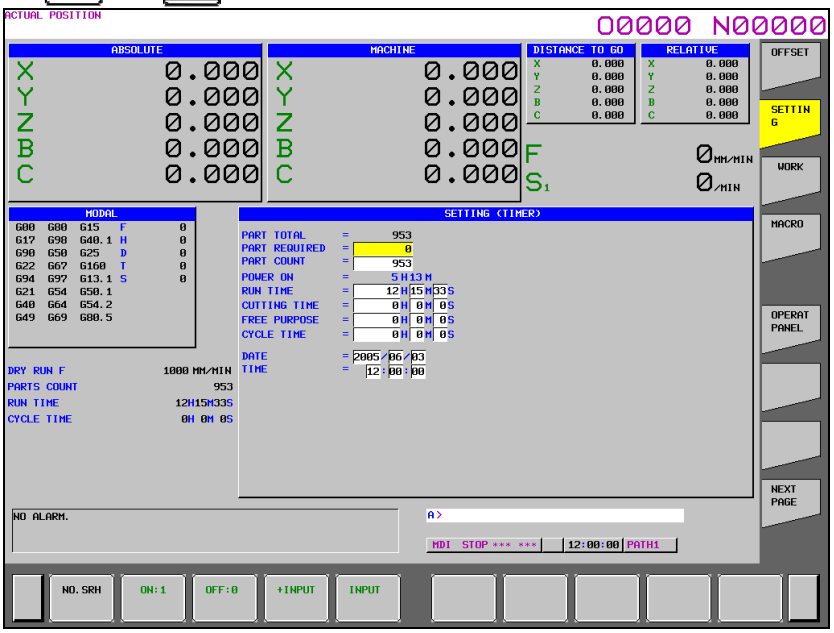


Fig. 12.3.21 (a) SETTING (TIMER) screen (15-inch display unit)

5 To set the number of parts required, move the cursor to PARTS REQUIRED and enter the number of parts to be machined.
6 To set the clock, move the cursor to DATE or TIME, enter a new date or time, then press horizontal soft key [INPUT].

Explanation

- PARTS TOTAL

This value is incremented by one when M02, M30, or an M code specified by parameter No. 6710 is executed. This value cannot be set on this screen. Set the value in parameter No. 6712.

- PARTS REQUIRED

It is used for setting the number of machined parts required.
When the "0" is set to it, there is no limitation to the number of parts.
Also, its setting can be made by the parameter No 6713.

- PARTS COUNT

This value is incremented by one when M02, M30, or an M code specified by parameter No. 6710 is executed. The value can also be set by parameter No. 6711. In general, this value is reset when it reaches the number of parts required. Refer to the manual issued by the machine tool builder for details.

- POWER ON

Displays the total time which the power is on. This value cannot be set on this screen but can be preset in parameter No. 6750.

- RUN TIME

Indicates the total run time during automatic operation, excluding the stop and feed hold time.
This value can be preset in parameter No. 6751 or No. 6752.

- CUTTING TIME

Displays the total time taken by cutting that involves cutting feed such as linear interpolation (G01) and circular interpolation (G02 or G03). This value can be preset in parameter No. 6753 or No. 6754.

- FREE PURPOSE

This value can be used, for example, as the total time during which coolant flows. Refer to the manual issued by the machine tool builder for details.

- CYCLE TIME

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- DATA and TIME

Displays the current date and time. The date and time can be set on this screen.

- Usage

When the command of M02 or M30 is executed, the total number of machined parts and the number of machined parts are incremented by one. Therefore, create the program so that M02 or M30 is executed every time the processing of one part is completed. Furthermore, if an M code set to the parameter No. 6710 is executed, counting is made in the similar manner. Also, it is possible to disable counting even if M02 or M30 is executed (bit 0 (PCM) of parameter No. 6700 is set to 1). For details, see the manual issued by machine tool builders.

Limitation

- Run time and part count settings

Negative value cannot be set. Also, the setting of "M" and "S" of run time is valid from 0 to 59.
Negative value may not be set to the total number of machined parts.

- Time settings

Neither negative value nor the value exceeding the value in the following table can be set.

Table 12.3.21 (a)

Item	Maximum value	Item	Maximum value
Year	2096	Hour	23
Month	12	Minute	59
Day	31	Second	59

12.3.22 Displaying and Setting the Workpiece Origin Offset Value (15/19-inch Display Unit)

Displays the workpiece origin offset for each workpiece coordinate system (G54 to G59, G54.1 P1 to G54.1 P48 and G54.1 P1 to G54.1 P300) and external workpiece origin offset.
The workpiece origin offset value and external workpiece origin offset value can be set on this screen.

Procedure for displaying and setting the workpiece origin offset value

Procedure

1 Press function key .

2 Press vertical soft key [WORK].
The workpiece coordinate system setting screen is displayed.

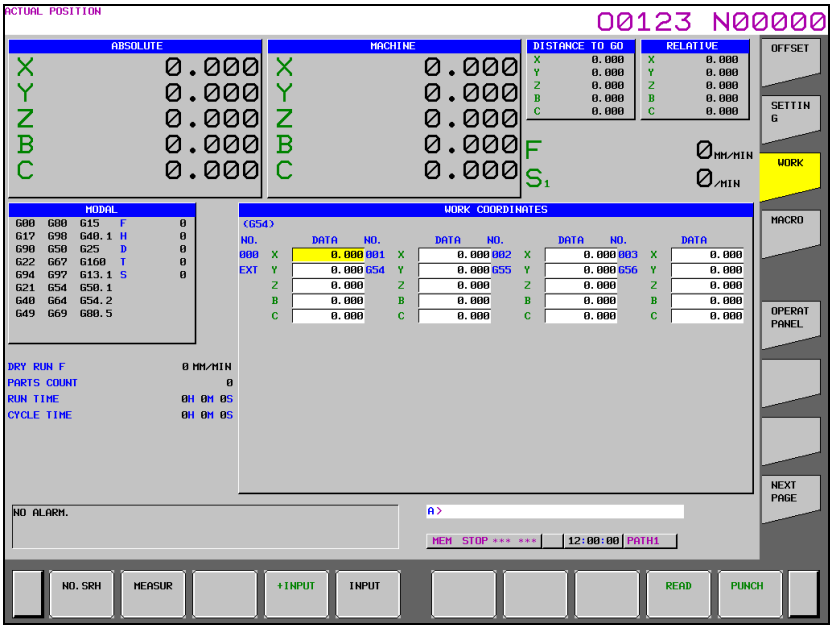


Fig. 12.3.22 (a) WORK COORDINATES screen (15-inch display unit)

3 The screen for displaying the workpiece origin offset values consists of two or more pages.
Display a desired page in either of the following two ways:

- Press the page key  or .
- Enter the workpiece coordinate system number (0 : external workpiece origin offset, 1 to 6: workpiece coordinate systems G54 to G59, P1 to P48 : workpiece coordinate systems G54.1 P1 to G54.1 P48, P1 to P300 : workpiece coordinate systems G54.1 P1 to G54.1 P300) and press horizontal soft key [NO.SRH].

4 Turn off the data protection key to enable writing.

5 Move the cursor to the workpiece origin offset to be changed.

6 Enter a desired value by pressing numeric keys, then press horizontal soft key [INPUT]. The entered value is specified in the workpiece origin offset value. Or, by entering a desired value with numeric keys and pressing horizontal soft key [+INPUT], the entered value can be added to the previous offset value.

7 Repeat steps 5 and 6 to change other offset values.

8 Turn on the data protection key to disable writing.

12.3.23 Direct Input of Workpiece Origin Offset Value Measured (15/19-inch Display Unit)

This function is used to compensate for the difference between the programmed workpiece coordinate system and the actual workpiece coordinate system. The measured offset for the origin of the workpiece coordinate system can be input on the screen such that the command values match the actual dimensions. Selecting the new coordinate system matches the programmed coordinate system with the actual coordinate system.

Procedure for direct input of workpiece origin offset value measured

Procedure

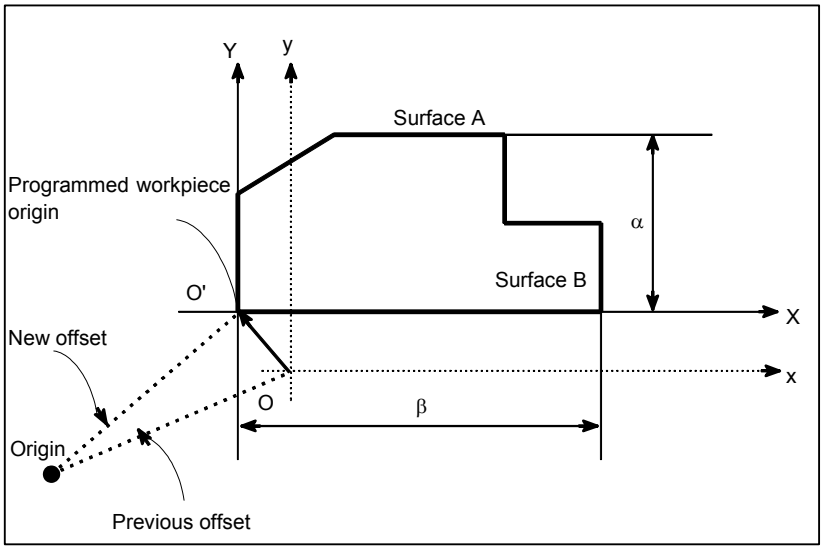


Fig. 12.3.23 (a)

1 When the workpiece is shaped as shown Fig. 12.3.23 (a), position the reference tool manually until it touches surface A of the workpiece.
2 Retract the tool without changing the Y coordinate.
3 Measure distance α between surface A and the programmed origin of the workpiece coordinate system as shown Fig. 12.3.23 (a).
4 Press function key .
5 To display the WORK COORDINATES screen, press vertical soft key [WORK].

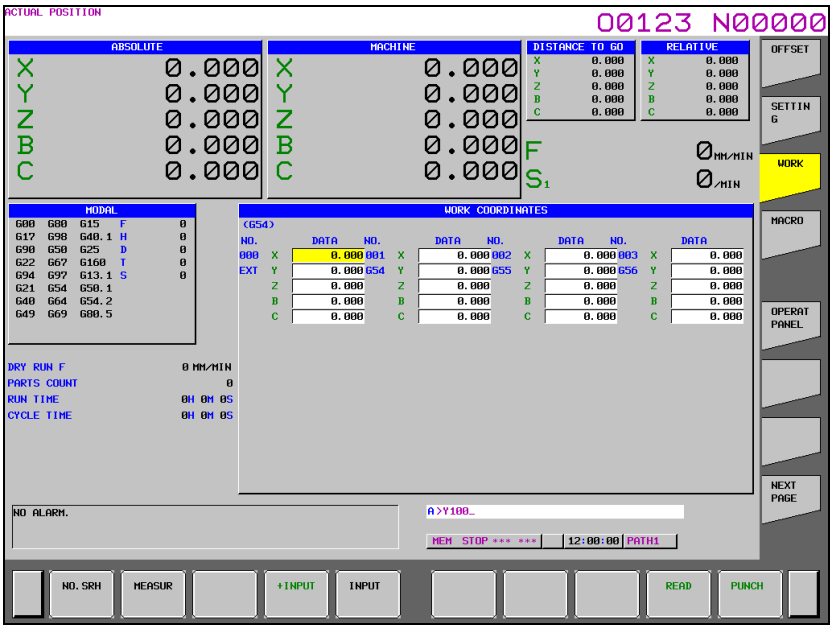


Fig. 12.3.23 (b) WORK COORDINATES screen (15-inch display unit)

6 Position the cursor to the workpiece origin offset value to be set.
7 Press the address key for the axis along which the offset is to be set (Y-axis in this example).
8 Enter the measured value (α) then press horizontal soft key [MEASUR].
9 Move the reference tool manually until it touches surface B of the workpiece.
10 Retract the tool without changing the X coordinate.
11 Enter the measured value of distance (β) to surface B as an X value as with steps 7 and 8.

Limitation

- Consecutive input

Offsets for two or more axes cannot be input at the same time.

- During program execution

This function cannot be used while a program is being executed.

12.3.24 Displaying and Setting Custom Macro Common Variables (15/19-inch Display Unit)

Displays common variables (#100 to #149 or #100 to #199, and #500 to #531 or #500 to #999) on the screen.
The values for variables can be set on this screen.
Relative coordinates can also be set to variables.

Procedure for displaying and setting custom macro common variables

Procedure

1 Press function key .
2 Press vertical soft key [MACRO].
The screen in the Fig. 12.3.24 (a) is displayed.

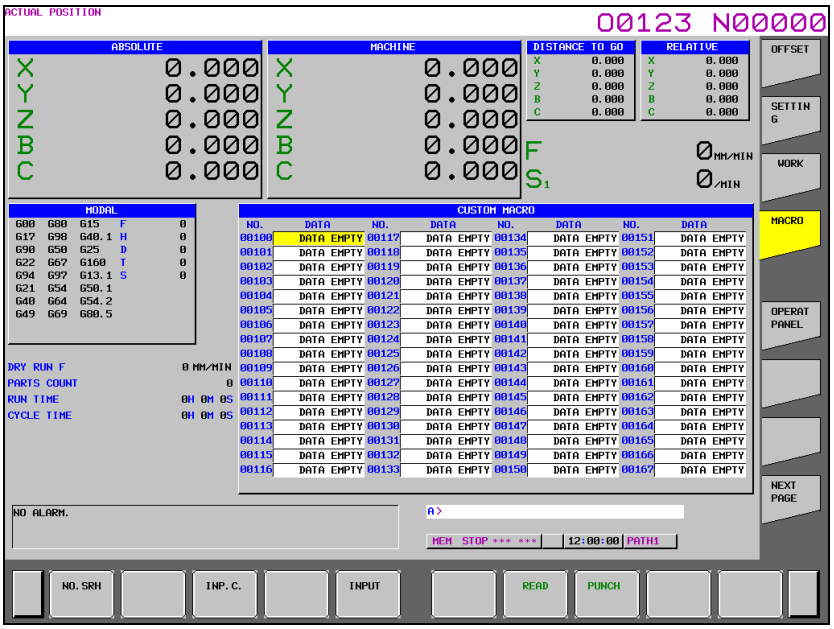


Fig. 12.3.24 (a) CUSTOM MACRO screen (15-inch display unit)

3 Move the cursor to the variable number to set using either of the following methods:
- Enter the variable number and press horizontal soft key [NO.SRH].
- Move the cursor to the variable number to set by pressing page keys PAGE and/or PAGE and cursor keys ↑, ↓, ←, and/or →.

4 Enter data with numeric keys and press horizontal soft key [INPUT].

5 To set a relative coordinate in a variable, press address key X, Y, or Z, then press horizontal soft key [INP.C.].

6 To set a blank in a variable, just press horizontal soft key [INPUT]. The value field for the variable becomes blank.

Explanation

If the value of a variable produced by an operation is not displayable, an indication below is provided.
When the significant number of digits is 12 (with bit 0 (F16) of parameter No. 6008 set to 0):

Variable value range	Variable value indication
0 < Variable value < +0.00000000001	+Underflow
0 > Variable value > -0.00000000001	-Underflow
Variable value > 999999999999	+Overflow
Variable value < -999999999999	-Overflow

When the significant number of digits is 8 (with bit 0 (F16) of parameter No. 6008 set to 1):

Variable value range	Variable value indication
0 < Variable value < +0.0000001	+Underflow
0 > Variable value > -0.0000001	-Underflow
Variable value > 99999999	+Overflow
Variable value < -99999999	-Overflow

12.3.25 Displaying and Setting Real Time Custom Macro Data (15/19-inch Display Unit)

Real time macro variables (RTM variables) are dedicated to real time custom macros.
RTM variables are divided into temporary real time macro variables (temporary RTM variables) and permanent real time macro variables (permanent RTM variables).
The values of permanent RTM variables are kept stored after the power is turned off.
Temporary RTM variables are cleared to 0 when the power is turned off.

System variables (DI/DO variables) dedicated to real time custom macros are used to read and write PMC interface signals. Data can be read and written in bit and byte units. Before writing a signal, release the protection of the corresponding variable on the PMC signal protect screen.

Displaying and setting real time custom macro variables

Procedure

1 Press function key [SYSTEM].

2 Press vertical soft key [NEXT PAGE] several times and then vertical soft key [R.TIME MACRO]. The screen shown below appears.

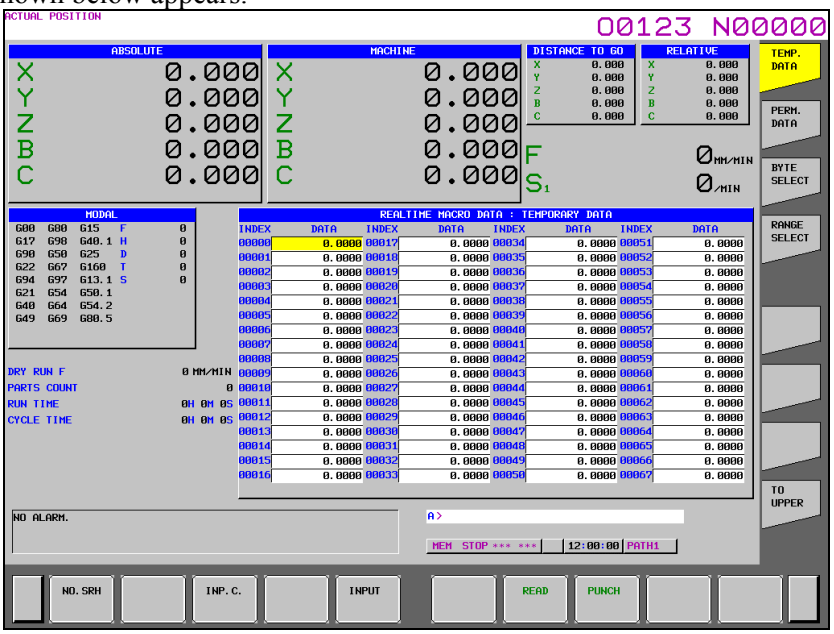


Fig. 12.3.25 (a) REAL TIME MACRO screen (15-inch display unit)

3 To display or set real time custom macro variables of which values are not stored at power-off, press vertical soft key [TEMP. DATA].

4 To display or set real time custom macro variables of which values are stored at power-off, press vertical soft key [PERM. DATA].

5 Move the cursor to the number of a real time custom macro variable you want to set using either of the following methods:

- Enter the number of a real time custom macro variable and press horizontal soft key [NO. SRH].
- Move the cursor to the number of a real time custom macro variable you want to set by pressing page keys [PAGE↑] and/or [PAGE↓] and cursor keys [↑], [↓], [←], and/or [→].

6 Enter data.

Displaying and setting DI/DO variables

Procedure

For setting in byte units:

1 Press function key SYSTEM.

2 Press vertical soft key [NEXT PAGE] several times and then vertical soft key [R.TIME MACRO].

3 Press vertical soft key [BYTE SELECT]. The following screen appears:

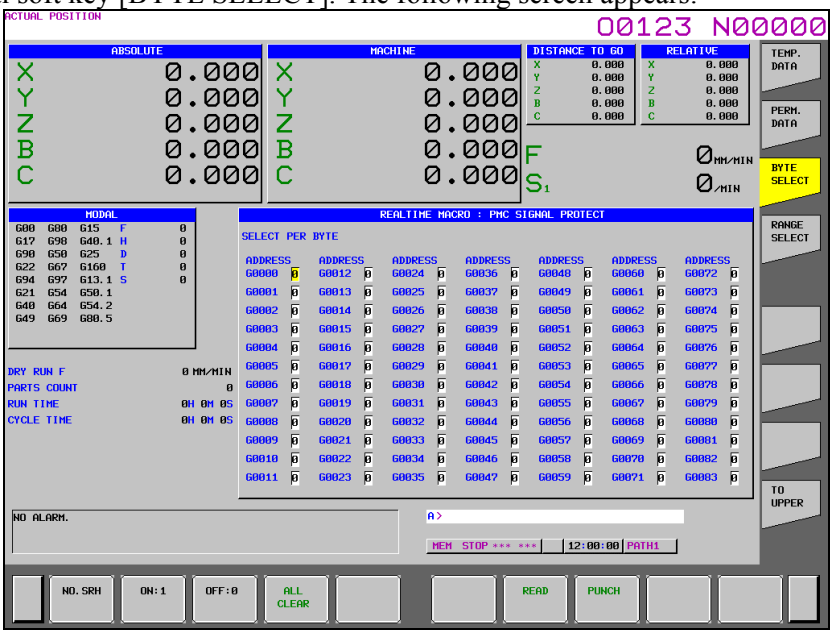


Fig. 12.3.25 (b) BYTE SELECT screen (15-inch display unit)

4 Move the cursor to the number of a DI/DO variable you want to set using either of the following methods:

- Enter the number and press horizontal soft key [NO. SRH].
- Move the cursor to a desired number by pressing page keys PAGE and/or PAGE and cursor keys , , , and/or .

5 Enter data.

12.3.26 Displaying and Setting the Software Operator's Panel (15/19-inch Display Unit)

Operations on the MDI unit can substitute for the functions of switches on the machine operator's panel. This means that a mode selection, jog feed override selection, and so forth can be made with operations on the MDI unit, eliminating the need to use the corresponding switches on the machine operator's panel. Jog feed can be performed using numeric keys.

Procedure for displaying and setting the software operator's panel

Procedure

1 Press function key OFS SET.

2 Press vertical soft key [NEXT PAGE] and then vertical soft key [OPERAT PANEL].

3 The screen consists of several pages. Press page key PAGE or PAGE until the desired screen is displayed.

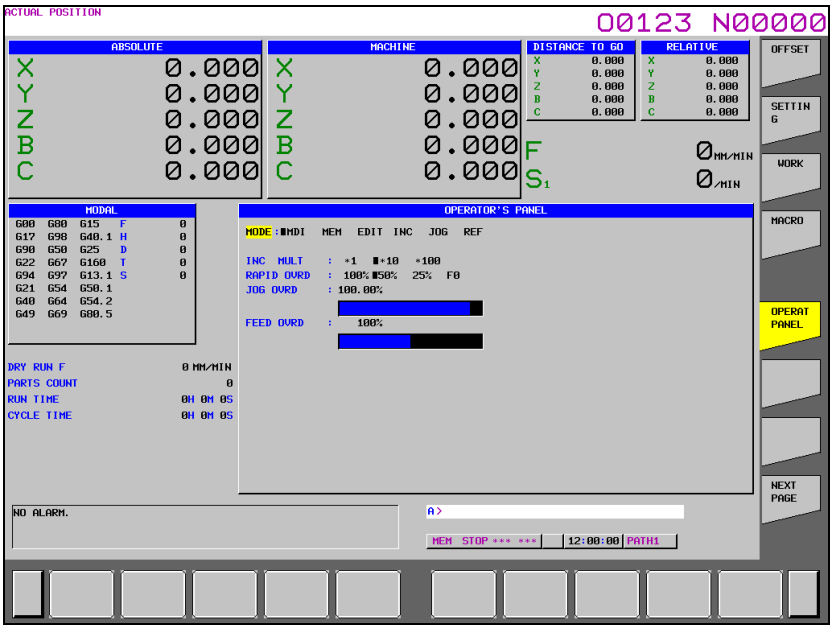


Fig. 12.3.26 (a) Without the manual handle feed function (15-inch display unit)

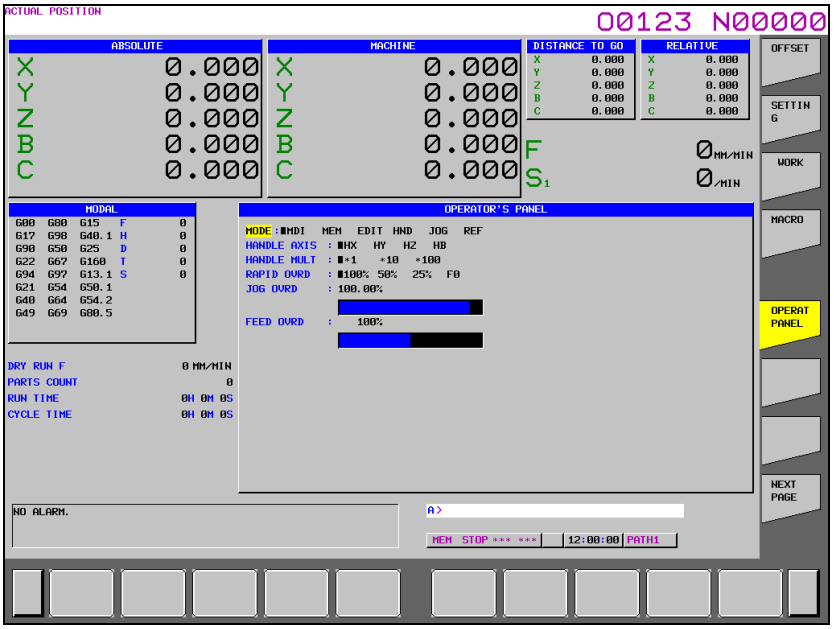


Fig. 12.3.26 (b) With the manual handle feed function (15-inch display unit)

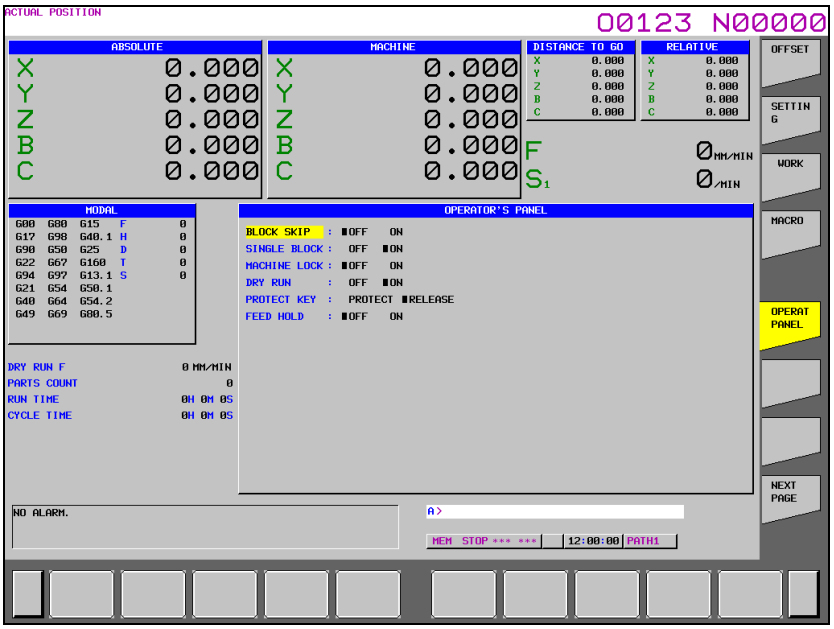


Fig. 12.3.26 (c) (15-inch display unit)

4 Move the cursor to the desired switch by pressing cursor key ↑ or ↓.

5 Push the cursor key ← or → to match the mark ■ to an arbitrary position and set the desired condition.

6 Press one of the following arrow keys (Fig. 12.3.26 (d)) to perform jog feed. Press the JOG 5 key together with an arrow key to perform jog rapid traverse.

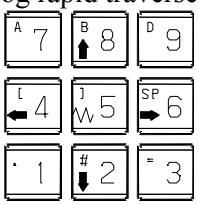


Fig. 12.3.26 (d) MDI arrow keys

Explanation

- Valid operations

The valid operations on the software operator's panel are shown below. Whether to use each group can be chosen using parameter No. 7200. Those groups that are not used are not displayed on the software operator's panel.

Group1 : Mode selection
Group2 : Selection of jog feed axis, Manual rapid traverse
Group3 : Selection of manual pulse generator feed axis, selection of manual pulse magnification
Group4 : Jog federate, federate override, rapid traverse override
Group5 : Optional block skip, single block, machine lock, dry run
Group6 : Protect key
Group7 : Feed hold

- Screens on which jog feed is valid

When the LCD indicates other than the software operator's panel screen and self-diagnosis screen, jog feed is not conducted even if the arrow key is pushed.

- **Jog feed and arrow keys**

The feed axis and direction corresponding to the arrow keys can be set with parameters (Nos. 7210 to 7217).

- **General purpose switches**

For the meanings of these switches, refer to the manual issued by machine tool builder.

12.3.27 Setting and Displaying Tool Management Data (15/19-inch Display Unit)

The tool management function totally manages tool information including tool offsets and tool life information.
This function provides a magazine screen and tool management screen. This subsection explains how to set these screens.

12.3.27.1 Displaying and setting magazine screen (15-inch display unit)

Procedure

1 Press function key .

2 Press vertical soft key [NEXT PAGE] several times and then vertical soft key [TOOL MANAGER]. The screen shown below appears.

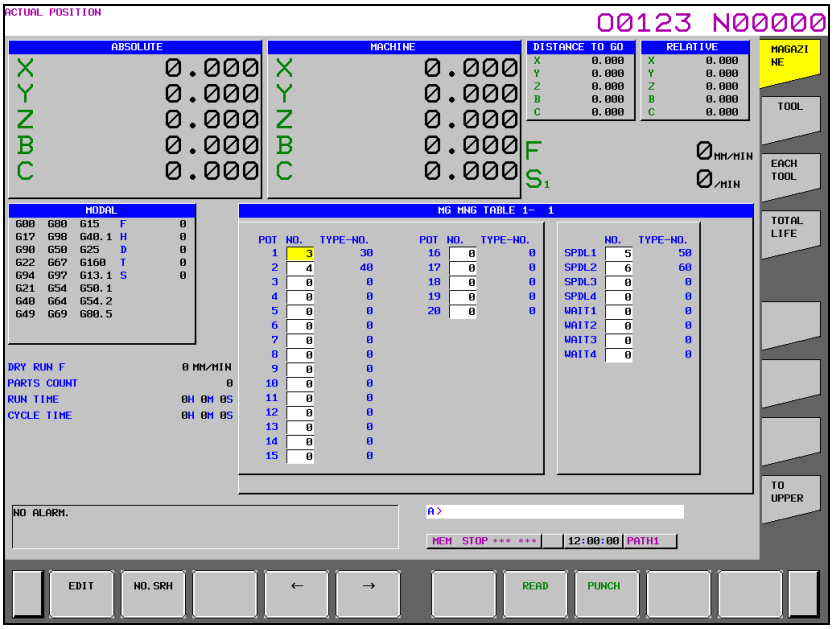


Fig. 12.3.27.1 (a)) Tool management magazine screen (15-inch display unit)

3 By using the page keys, cursor keys, and horizontal soft keys [←] and [→], move the cursor to the pot No. position of the magazine for which you want to set or modify data.
Alternatively, type the number of the tool management data you wan to set or modify, and press horizontal soft key [NO. SRH].

4 Press horizontal soft key [EDIT].

5 To set the tool management data number of a pot, type the tool management data number, then press MDI key .

To delete the tool management data number set for a pot, follow the steps below.
<1> Press horizontal soft key [ERASE].
<2> Press horizontal soft key [CURDAT]. To delete the tool management data numbers registered for all pots, press horizontal soft key [ALL].
<3> Press horizontal soft key [EXEC]. Alternatively, type 0.

6 To end the edit operation, press horizontal soft key [EXIT].
This returns the screen display to the conventional tool management screen.

Explanation

- Another method

Magazine data can be input/output also by using external I/O devices.
See III-8, "DATA I/O".

- Displayed data

POT : As many pots as specified in parameter No. 13222, 13227, 13232, or 13237 (settable for each magazine) are displayed. The starting pot number can be set for each magazine in parameter No. 13223, 13228, 13233, or 13238.

NO. : Tool management data numbers are displayed.
In the following cases, tools cannot be set in magazines:
1. A tool management data number beyond the range from 0 to (the number of valid pairs in parameter No. 13220) is set.
2. Tool management data is invalid (bit 1 of tool information is 0).
3. The tool management data number to be set is already set for another pot.

TYPE NO. : Tool type numbers corresponding to tool management data numbers are displayed.
Values cannot be set on this screen.

SPINDLE : The tool management data numbers and tool type numbers of spindle positions are displayed.

WAIT : The tool management data numbers and tool type numbers of wait positions are displayed.

- Edit mode

To edit data, press horizontal soft key [EDIT] to enter tool management data edit mode.

- Tool management data number

Tool management data numbers can optionally be extended to 64, 240, or 1000 pairs. In addition, the number of tools used can be set in parameter No. 13220.

- Display of the spindle positions and wait positions

The spindle position and wait position can be set for each path.

12.3.27.2 Displaying and setting tool management screen (15-inch display unit)

Procedure

1 Press function key .
2 Press vertical soft key [NEXT PAGE] several times and then vertical soft key [TOOL MANAGER].
3 Press vertical soft key [TOOL].

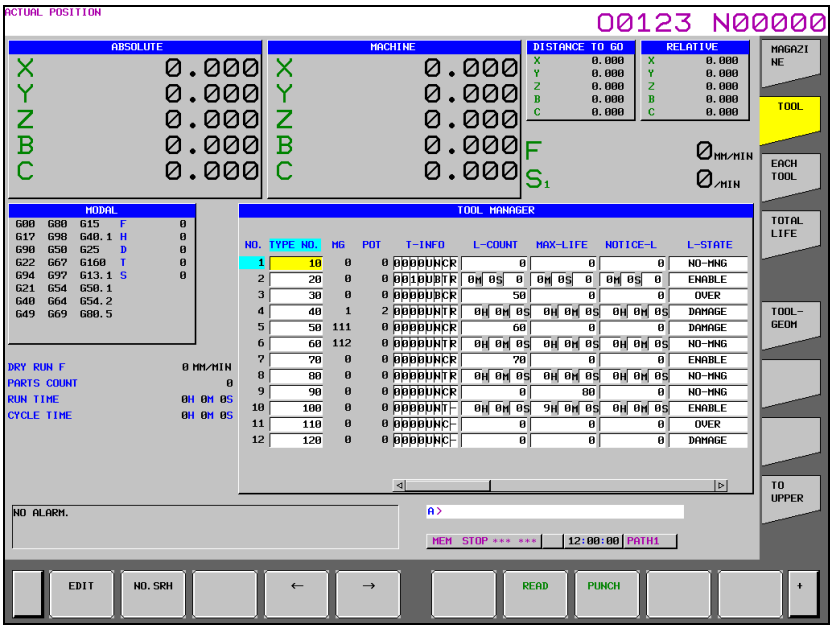


Fig. 12.3.27.2 (a) Tool management data screen (15-inch display unit)

4 By using the page keys, cursor keys, and horizontal soft keys [←] and [→], move the cursor to the position of the tool information of the tool number for which you want to set or modify data.
Alternatively, type the tool number of the data you want to set or modify, and press horizontal soft key [NO. SRH].

5 Press horizontal soft key [EDIT].

6 To set tool data, type a desired value, then press MDI key 

.

To delete tool information, follow the steps below.

<1> In step 4, move the cursor to the tool information you want to erase.
<2> Press horizontal soft key [ERASE].
<3> Press horizontal soft key [CURDAT]. To delete entire tool information, press horizontal soft key [ALL].
<4> Press horizontal soft key [EXEC].

7 To end the edit operation, press horizontal soft key [EXIT].
This returns the screen display to the conventional tool management screen.

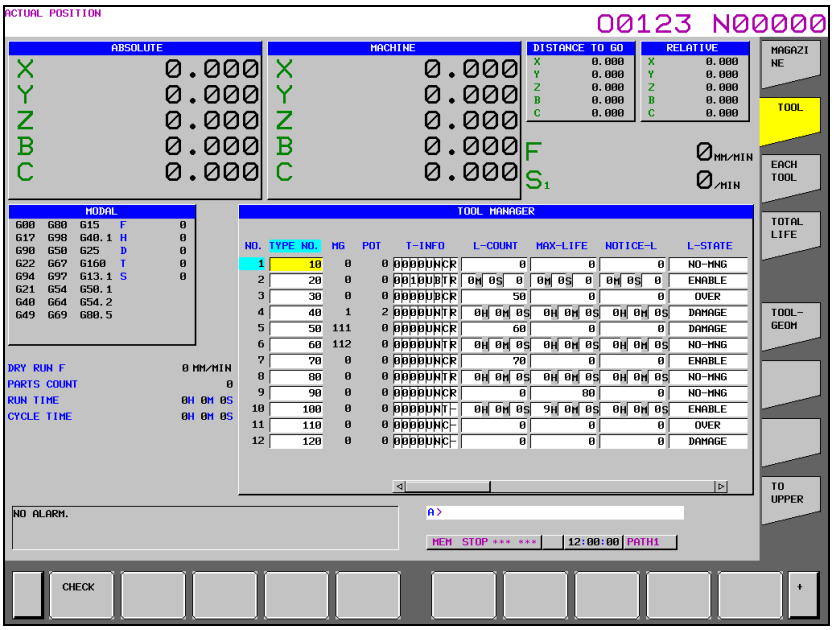


Fig. 12.3.27.2 (b) Tool management data screen (check function) (15-inch display unit)

8 When horizontal soft key [CHECK] is pressed, if there are tools with the same number but with different count types (count and time), the cursor moves to the tool type number of the smallest tool management number in the tool type numbers and a warning message appears.
The warning messages issued by the check function are shown below.
<1> For the same type of tools with different count types
L-COUNT TYPE MISMATCH:XXXXXXXX
<2> For the same type of tools with the same count type
L-COUNT TYPE MATCH

Explanation

- Another method

Tool management data, customize data, and names set for tool states can be input/output also by using external I/O devices.
See III-8, "DATA I/O".

- Displayed information

- Life information

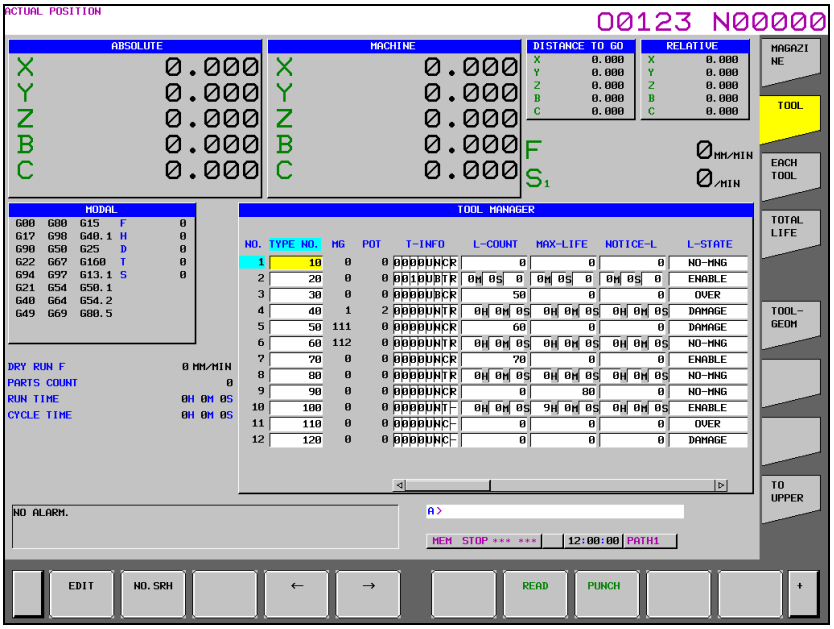


Fig. 12.3.27.2 (c) Tool management data life status screen(15-inch display unit)

NO. : Tool management data numbers are displayed. These numbers can be displayed but cannot be set. The tool management data number of edited data is kept blinking until tool management data edit mode ends. Tool management data numbers set or deleted on the magazine screen also blink on the tool management screen.

TYPE NO. : Tool type numbers are displayed.
A value from 0 to 99,999,999 can be set.

MG : The magazine number assigned to each tool is indicated. These numbers can be displayed but cannot be set.

POT : The pot number assigned to each tool is indicated. These numbers can be displayed but cannot be set.

T-INFO : The following five types of information are displayed sequentially from the right:

- Tool management data: Valid (R)/invalid (-))
- Life count type: Time (T)/count (C)
- Tool type: Oversize tool (B)/normal tool (N)
- Data access: Locked (L)/unlocked (U)
- When a tool is not under life management:
  Included in the tools to be searched (1)/not included in the tools to be searched (0)

NOTE
1 The tool types and data access information vary depending on the specifications defined by the machine tool builder.
2 The same type of tools must have the same life count type.

L-COUNT : The number of use times/use period of time of each tool is indicated.
Up to 99,999,999 times or 999 hours 59 minutes 59 seconds can be set.

MAX-LIFE : The maximum life value of each tool is indicated.
Up to 99,999,999 times or 999 hours 59 minutes 59 seconds can be set.

NOTICE-L : Noticed life value of a tool
Up to 99,999,999 times or 999 hours 59 minutes 59 seconds can be set.

L-STATE : Current tool state
One of the four states, including invalid (0), present (1, 2), not present (3), and broken (4), is indicated. The numbers in parentheses are data values used when these states are input in MDI.

- Spindle speed/feedrate

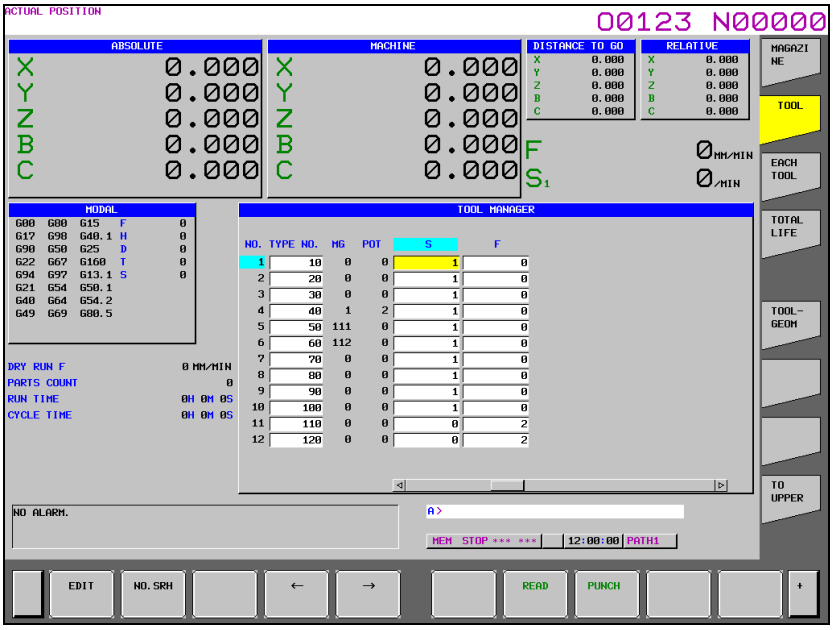


Fig. 12.3.27.2 (d) Tool management data spindle speed/feedrate screen (15-inch display unit)

S : Spindle speed.
A value from 0 to 99,999 can be set.
F : Feedrate.
A value from 0 to 99,999,999 can be set.

- Tool offset information

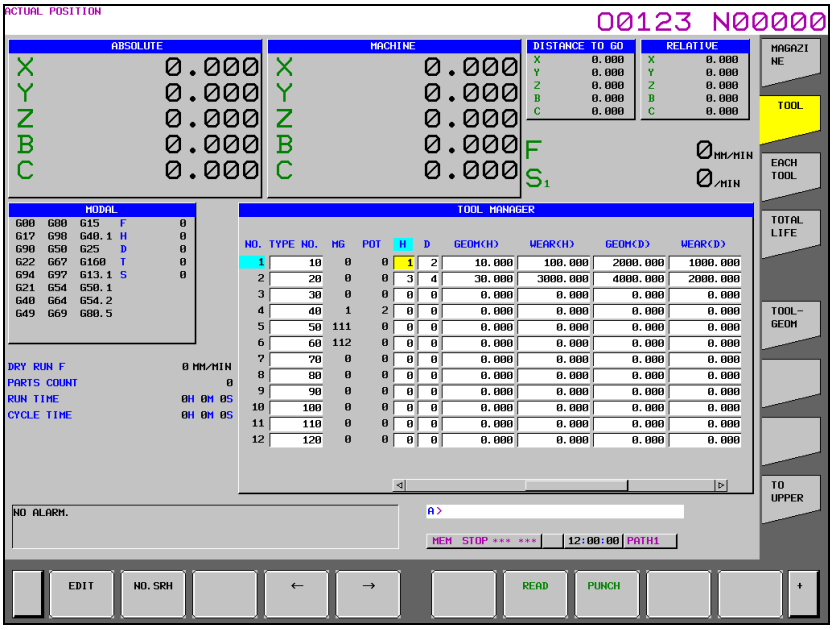


Fig. 12.3.27.2 (e) Tool management data tool offset screen (15-inch display unit)

H : Tool length compensation number (for machining center systems only).
A value from 0 to 999 can be set.

D : Cutter compensation number (for machining center systems only).
A value from 0 to 999 can be set.
TG : Tool geometry compensation number (for lathe systems only).
A value from 0 to 999 can be set.
TW : Tool wear compensation number (for lathe systems only).
A value from 0 to 999 can be set.

The displayed offset data is determined depending on the option configuration and parameter settings (bits 1(DCR), 2(DOY), 3(DOB), 4(DO2), 6(DOT), and 7(DOM) of parameter No. 13202).

- Customize information

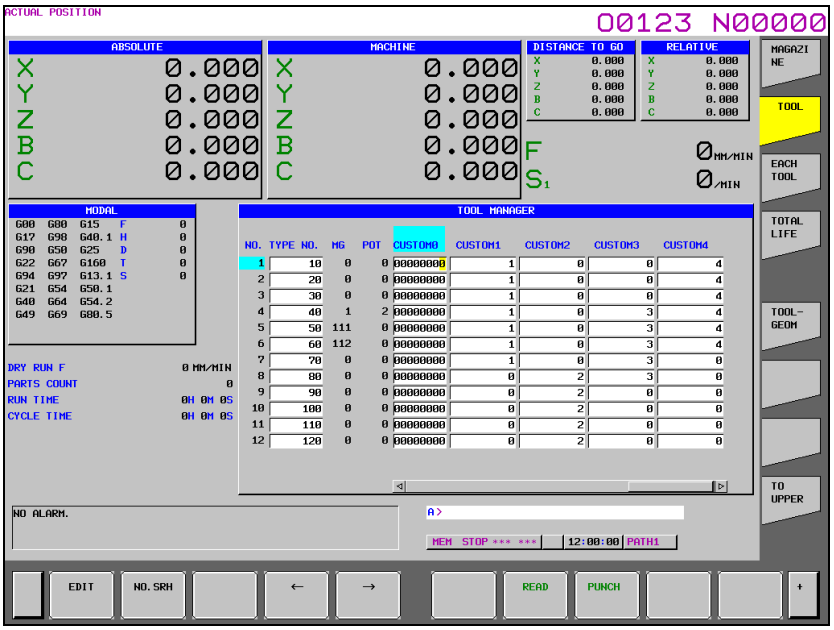


Fig. 12.3.27.2 (f) Tool management data customize data screen (15-inch display unit)

CUSTOM0 : Bit-type customize information.
For each bit, 1 or 0 can be input.
CUSTOM1 to 4 : Customize information. Any value from -99,999,999 to 99,999,999 can be set.
CUSTOM5 to 20 : Customize information. These items are displayed only when customize data extension option (5 to 20) of the tool management function is enabled. Any value from -99,999,999 to 99,999,999 can be set.
CUSTOM21 to 40 : Customize information. These items are displayed only when customize data extension option (5 to 40) of the tool management function is enabled. Any value from -99,999,999 to 99,999,999 can be set.

- Tool management data number

Tool management data numbers can optionally be extended to 64, 240, or 1000 pairs. The number of tools used can be set in parameter No. 13220.

- Edit mode

To edit data, press horizontal soft key [EDIT] to enter tool management data edit mode.

- Life count override

When the period of time is set as the tool life count type, the life count can be overridden if bit 2 (LVF) of parameter No. 6801 is set to 1. Specify an override value by using a switch on the machine operator's panel.
Example)

When cutting is performed for 10 minutes with an override of 0.1, one minute is counted in the tool life counter.

- Tool management extension function

When tool management extension functions are enabled, you can use the following functions in addition to the tool management functions:

- A value with a decimal point can be set as customize data.
  The maximum number of decimal places can be set to 7 by G10 data input and file input.
- Various types of tool management data can be protected using KEY signals.
  An attempt to enter a value to a protected data causes a warning message “WRITE PROTECT.”
- Tool life count period selection
  Bit 5 of tool information is used to make a life count period selection.

Item		Description
Data length		1 byte (flag data)
#5	REV	0: A life count period of 1 sec is used. 1: A life count period of 8 msec is used.

Range of count is as follows.

1sec : 0 to 3,599,999 seconds (999 hours 59 minutes 59 seconds)
8msec : 0 to 3,599,992 ms (59 minutes 59 seconds 992 milliseconds)

12.3.27.3 Each tool data screen (15-inch display unit)

Each tool data screen

Procedure

1 Press function key .

2 Press vertical soft key [NEXT PAGE] several times and then vertical soft key [TOOL MANAGER].

3 Press vertical soft key [EACH TOOL]. The each tool data screen appears.

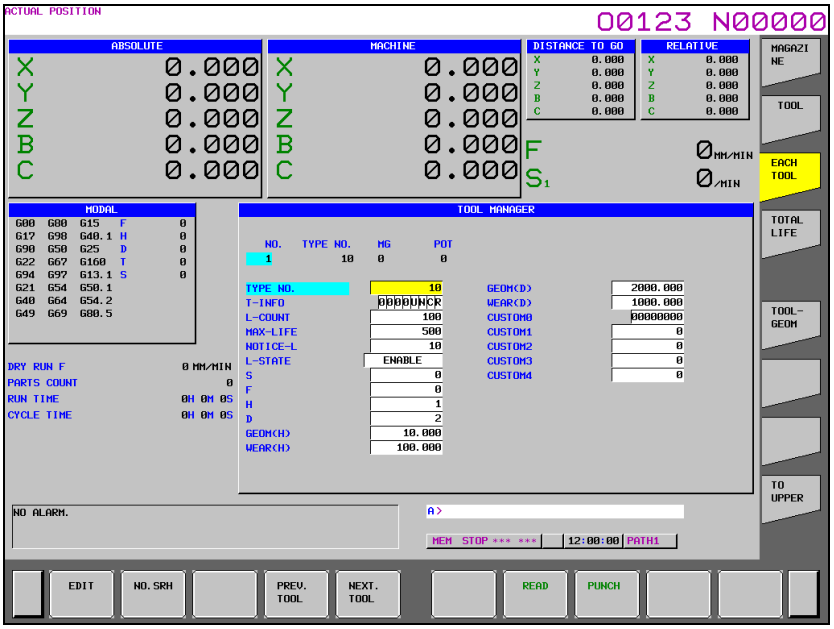


Fig. 12.3.27.3 (a) Each tool data screen (15-inch display unit)

Explanation

- Header

The following four data items are displayed: NO., TYPE NO., MG, and POT.
When the data table of a tool extends over two or more pages, the same header is displayed on these pages.

- Data table

The data table shows data items related to a tool at a time.
The data items are displayed from the upper left to the lower left, the upper right, and the lower right in ascending order of screen display position numbers specified using the customize function. The number of digits displayed for one data item is fixed to 11.
Up to 24 data items are displayed on one page. (12 data items × 2 columns)
When more than 24 data items are set for a tool, the 25th and subsequent data items are displayed on the next page. (Up to three pages)
When a data item is set as a screen element of the tool management data screen twice or more using the tool management data display customize function (one of the tool management extension functions), only the data item with the smaller display position number is displayed. When no value is set for a data item, the field for the data item is not displayed and the next data item is displayed.

Key operations

- MDI key operations

Numeral keys Inputs a numeric value.



Displays the previous page for the same tool.



Displays the next page for the same tool.



Moves the cursor to the previous data item.

When the cursor is on the top row of the data table, it moves to the bottom row of the left column. When the cursor is on the first data item, it moves to the last data item.



Moves the cursor to the next data item.

When the cursor is on the bottom row of the data table, it moves to the top row of the right column. When the cursor is on the last data item, it moves to the first data item.



Moves the cursor left on the screen.

When the cursor is on the left column of the data table, it moves to the right column on the row immediately above. When the cursor is on the first data item, it moves to the last data item.



Moves the cursor right on the screen.

When the cursor is on the right column of the data table, it moves to the left column on the row immediately below. When the cursor is on the last data item, it moves to the first data item.

- Soft key operations

Horizontal soft key [EDIT]
Enters the management data edit mode.
Horizontal soft key [NO.SRH]
Displays data with a tool management data number when pressed after the tool management data number is entered.
Horizontal soft key [PREV.TOOL]
Returns to the previous tool management data number.

Horizontal soft key [NEXT.TOOL]
Proceeds to the next tool management number.
Horizontal soft key [F INPUT]
Inputs data related to the tool management function.
Can be used only in the standard mode.
Requires placing the NC in the EDIT mode.
Horizontal soft key [F OUTPUT]
Writes data related to the tool management function.
Can be used only in the standard mode.
Requires placing the NC in the EDIT mode.

Operation in the management data edit mode

To edit data, press horizontal soft key [EDIT] to enter the management data edit mode.

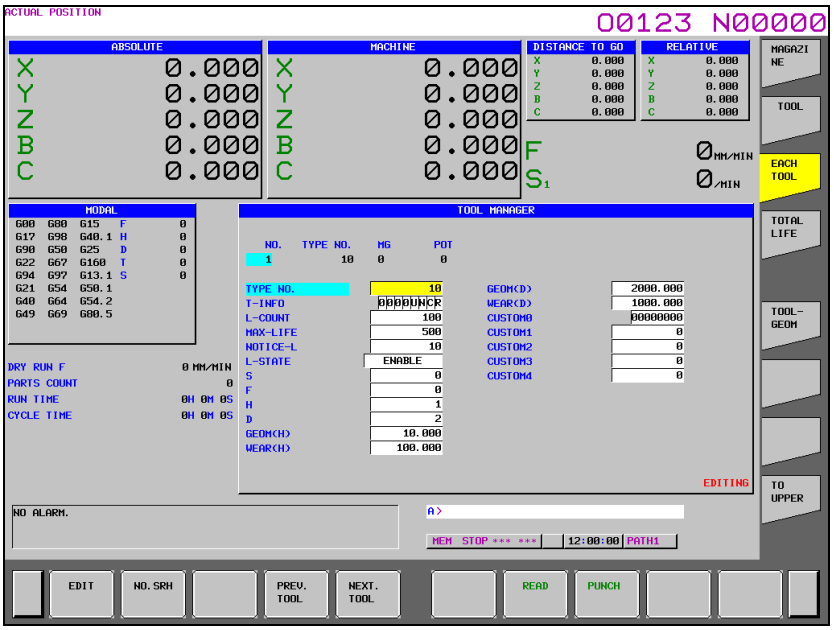


Fig. 12.3.27.3 (b) Each tool data editing screen (15-inch display unit)

In the management data edit mode, “EDITING” is displayed at the lower right of the screen.
In the management data edit mode, the above key operations (Fig. 12.3.27.3 (b)), in addition to those mentioned above except [F INPUT] and [F OUTPUT], can be performed.



Actually writes a value entered using numeric keys.

Horizontal soft key [END]
Ends the management data edit mode.

Operations for editing tool information are the same as those on the tool management data screen.

12.3.27.4 Displaying the total life of tools of the same type (15-inch display unit)

Total life data screen

Procedure

1 Press function key 

.

2 Press vertical soft key [NEXT PAGE] several times and then vertical soft key [TOOL MANAGER].

3 Press vertical soft key [TOTAL LIFE]. The total life data screen appears.
4 Using horizontal soft key [CHANGE] can switch total tool life data displays between specification by count and specification by duration.

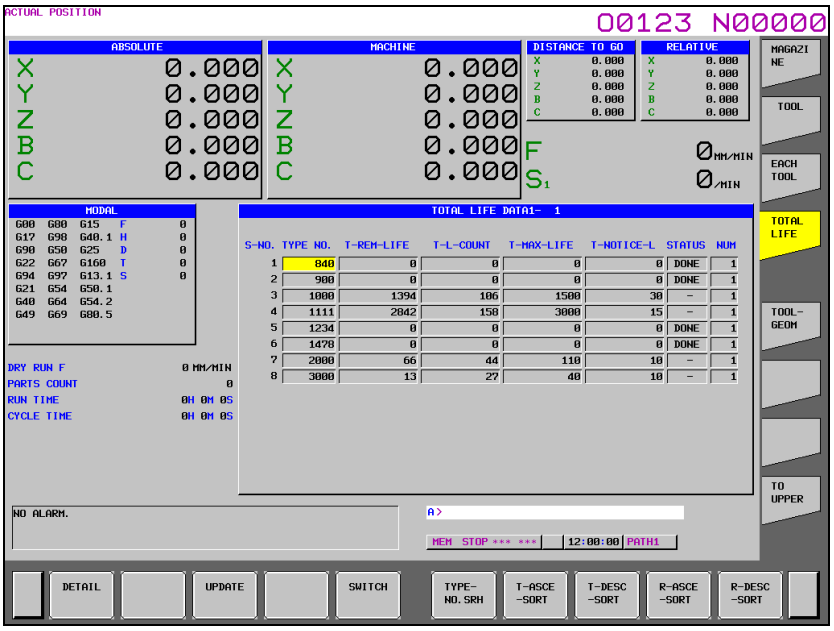


Fig. 12.3.27.4 (a) Count display (15-inch display unit)

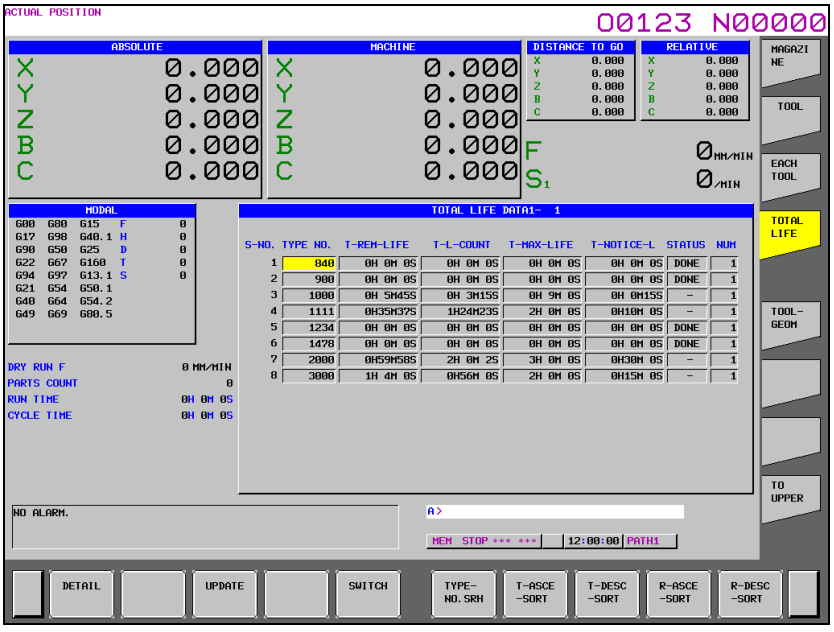


Fig. 12.3.27.4 (b) Time display (15-inch display unit)

Displayed information

S-NO. : Sequential number of each tool type
TYPE NO. : Tool type number
T-REM-LIFE : Total of remaining life values of tools with the same tool type number
T-L-COUNT : Total of used counts/times of tools with the same tool type number
T-MAX-LIFE : Total of maximum life values of tools with the same tool type number
T-NOTICE-L : Total of notice life values of tools with the same tool type number when the tool life arrival signal is to be output for each tool type
STATUS : Whether the tool life arrival signal has been output when the signal is to be output for each tool type

Either of the two states (UNDONE and DONE) is displayed.
NUM : Number of tools with the same tool type number

When bit 3 (ETE) of parameter No. 13200 is set to 0 and bit 2 (TRT) of parameter No. 13200 is set to 1, the tool life arrival signal is to be output for each tool type number and T-NOTICE-L and STATUS are displayed.

Key operations

- MDI key operations



Displays the previous page. The cursor moves to the last data item on that page.



Displays the next page. The cursor moves to the first data item on that page.


Moves the cursor up on the screen.


Moves the cursor down on the screen.


Disabled.


Disabled.

- Soft key operations

Horizontal soft key [DETAIL]	Displays the detailed life data screen.
Horizontal soft key [UPDATE]	Updates data displayed on the total life data screen.
Horizontal soft key [CHANGE]	Switch the display between the count and time types.
Horizontal soft key [T-TYPENO. SRH]	Moves the cursor to a tool type number when pressed after the number is entered.
Horizontal soft key [T-ASCE -SORT]	Sorts data in ascending order of tool type numbers.
Horizontal soft key [T-DESC -SORT]	Sorts data in descending order of tool type numbers.
Horizontal soft key [R-ASCE -SORT]	Sorts data in ascending order of remaining life values.
Horizontal soft key [R-DESC -SORT]	Sorts data in descending order of remaining life values.

NOTE
1 After horizontal soft key [T-ASCE -SORT], [T-DESC -SORT], [R-ASCE -SORT], or [R-DESC -SORT] is pressed, the cursor is positioned at the top of page 1 of the total life data screen.
2 When the power is turned on, data of the count counting type is displayed in ascending order of tool type numbers. When the display type is changed or data is sorted in a different order, the status is kept.
3 If horizontal soft key [DETAIL] is pressed when no data is displayed on the total life data screen, the warning message “NO DETAILED LIFE DATA SCREEN” is output.
4 If an unregistered tool type is specified and horizontal soft key [T-TYPENO. SRH] is pressed, the warning message “UNREGISTERED NUMBER” is output.

Detailed life data screen

Procedure

1 Press function key 

.
2 Press vertical soft key [NEXT PAGE] several times and then vertical soft key [TOOL MANAGER].
3 Press vertical soft key [TOTAL LIFE]. The total life data screen appears.
4 Selecting horizontal soft key [DETAIL] causes the detailed life data screen to appear.

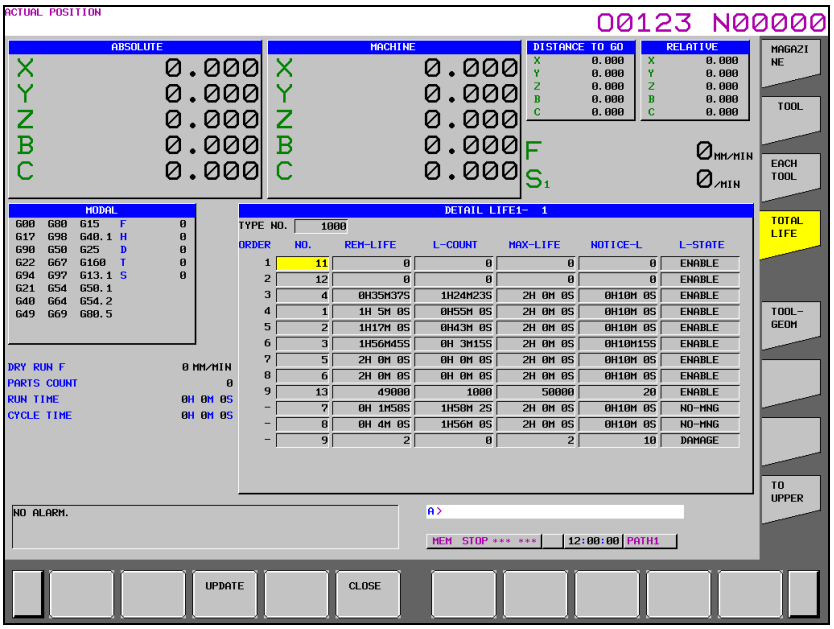


Fig. 12.3.27.4 (c) Detailed life data screen (15-inch display unit)

- Displayed information

TYPE NO. : Tool type number
ORDER : Sequential number in ascending order of remaining life times or the order in which the customize data is set.
NO. : Tool management data number
REM-LIFE : Remaining life value obtained by subtracting the life count value from the maximum life value
L-COUNT : Total used count or time of the tool
MAX-LIFE : Maximum life value of the tool
NOTICE-L : Notice life value of the tool
STATUS : Life status of the tool
One of the four states (NO-MNG (0), ENABLE (1, 2), NONE (3), and DAMAGED (4)) is displayed.

Key operations

- MDI key operations



Displays the previous page.



Displays the next page.


Moves the cursor up on the screen. The cursor moves to the last data item on that page.


Moves the cursor down on the screen. The cursor moves to the first data item on that page.


Disabled.


Disabled.

- Soft key operations

Horizontal soft key [UPDATE]
Updates data displayed on the detailed life data screen.
The cursor is positioned at the beginning of page 1 after this soft key is pressed.

Horizontal soft key [CLOSE]
Closes the detailed life data screen and returns to the total life data screen.

NOTE
When horizontal soft key [CLOSE] is pressed and the total life data screen is displayed again, the cursor on the total life screen data is positioned as follows.
- When the tools with the tool type number displayed when the detailed data screen is closed are registered as tool management data, the cursor on the total life screen is positioned at that tool type number.
- When the tools with the tool type number displayed when the detailed data screen is closed are not registered as tool management data, the cursor on the total life data screen is positioned at the first total life data.

12.3.27.5 Tool geometry data screen (15-inch display unit)

Tool geometry data screen

Procedure

1 Press function key .
2 Press vertical soft key [NEXT PAGE] several times and then vertical soft key [TOOL MANAGER].
3 Selecting vertical soft key [TOOL-GEOM] causes the tool geometry data screen to appear.

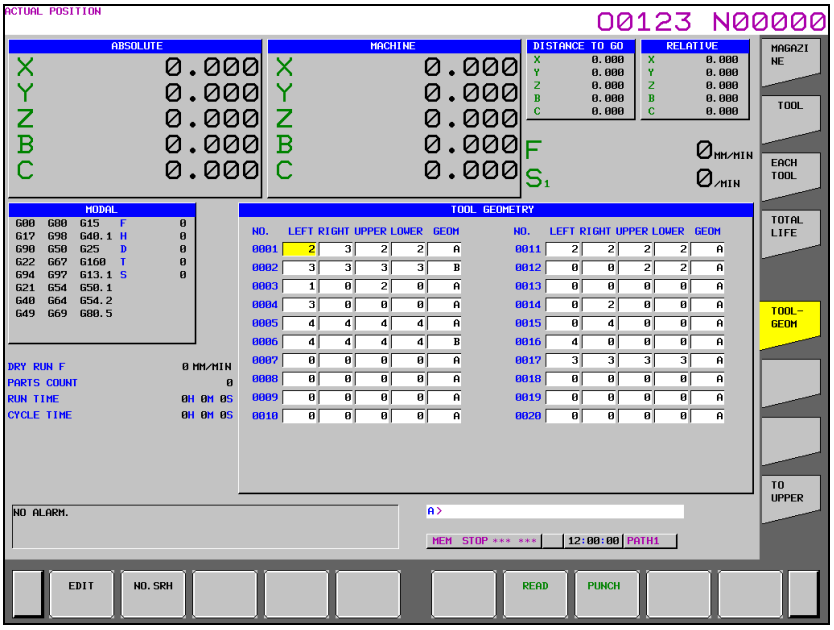


Fig. 12.3.27.5 (a) Tool geometry data screen (15-inch display unit)

- Displayed item

NO. : Tool geometry number
Up to 20 numbers can be displayed.
LEFT : Sets the number of pots on the left of the reference pot that are to be occupied.
A value between 0 and 4 can be set.
RIGHT : Sets the number of pots on the right of the reference pot that are to be occupied.
A value between 0 and 4 can be set.
UPPER : Sets the number of pots above the reference pot that are to be occupied.
A value between 0 and 4 can be set. (Use this item when the magazine is of the matrix type.)

LOWER : Sets the number of pots under the reference pot that are to be occupied.
A value between 0 and 4 can be set. (Use this item when the magazine is of the matrix type.)

Key operations

- Operations in the standard mode

MDI key operations

Numeral keys Inputs a numeric value.


Moves the cursor up on the screen.


Moves the cursor down on the screen.


Moves the cursor left on the screen.


Moves the cursor right on the screen.

Soft key operations

Horizontal soft key [NO.SRH]	Moves the cursor on a tool geometry number when pressed after the number is entered.
Horizontal soft key [EDIT]	Enters the data edit mode.
Horizontal soft key [F INPUT]	Inputs data related to the tool management functions. This key is available only in the standard mode. Put the NC in the EDIT mode.
Horizontal soft key [F OUTPUT]	Outputs data related to the tool management functions. This key is available only in the standard mode. Put the NC in the EDIT mode.

In the management data edit mode, the above key operations (Fig. 12.3.27.5 (a)), in addition to [NO.SRH] mentioned above, can be performed.

- Operations in the edit mode

To edit data, press horizontal soft key [EDIT] to enter the edit mode. In the edit mode, “EDITING” is displayed at the lower right of the screen.

MDI key operation



Actually writes a value entered using numeric keys.

Soft key operation

Horizontal soft key [END]
Ends the data edit mode.

Example

Set the edit mode. When the tool geometry with tool geometry number 1 occupies 1 pot in the left direction, 0.5 pots in the right direction, and 1.5 pots in the down direction, set data as shown in the Fig. 12.3.27.5 (b):

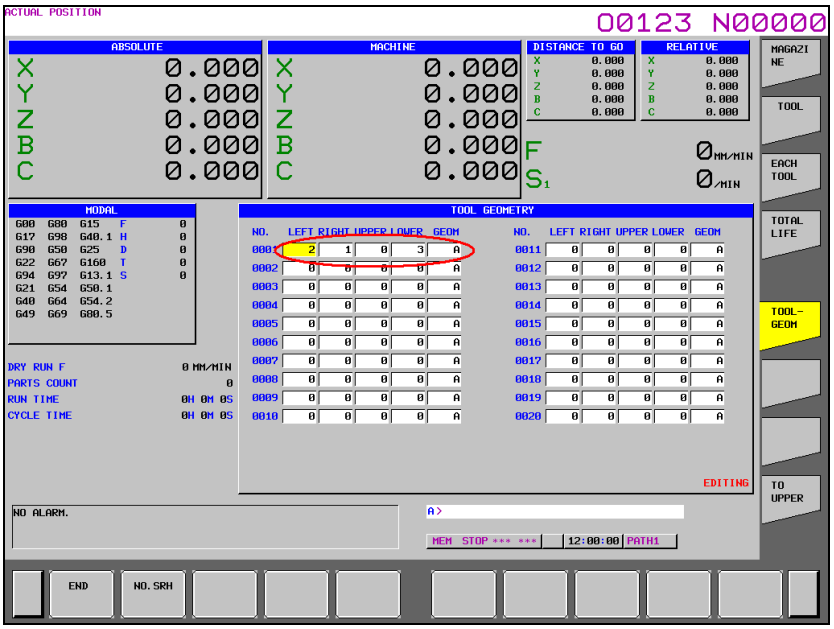


Fig. 12.3.27.5 (b) Example of setting data on the tool geometry data screen (15-inch display unit)

- Display of occupied pots in the magazine management table

Each pot occupied by a tool stored in another pot is indicated with an asterisk (*).

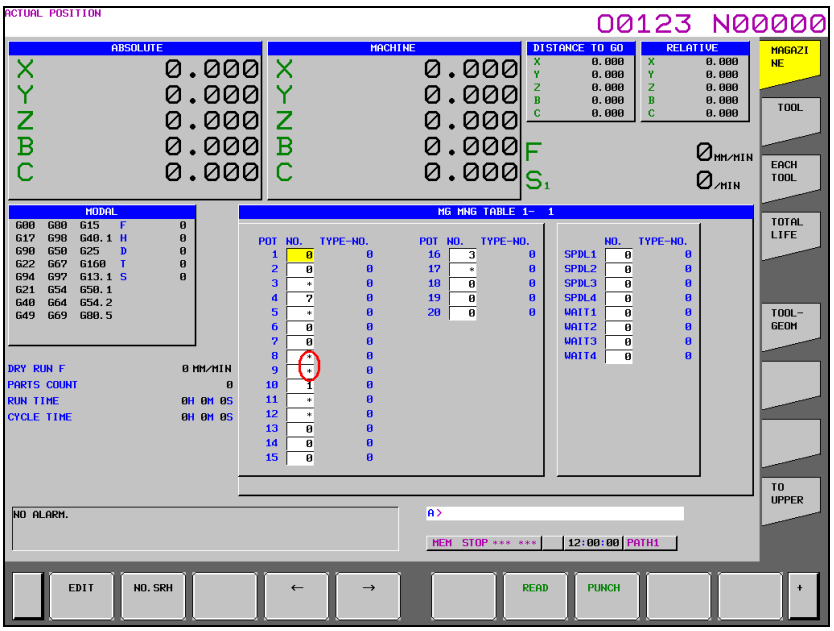


Fig. 12.3.27.5 (c) Magazine management table (15-inch display unit)

If a tool to be registered for a magazine is determined to interfere with another tool, the warning message "TOOL INTERFERENCE CHECK ERROR:xxxx,xxxx" is displayed. xxxx indicates the tool number of each of the two tools. If a tool is determined to interfere with multiple tools, only the first tool detected by the NC interference check is displayed.
If a tool is determined to interfere with the frame of a magazine, the tool number of the tool to be stored and "FRAME" are displayed.

- Search for an empty pot for a oversize tool

During the management data editing, a horizontal soft key for searching for empty oversize tool pots is displayed.

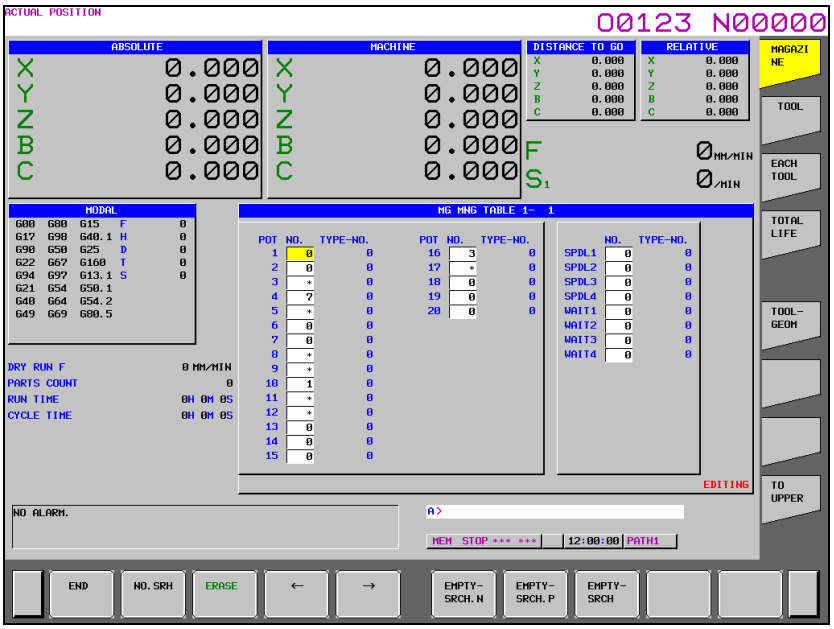


Fig. 12.3.27.5 (d) Searching for an empty pot for a oversize tool (15-inch display unit)

Enter the tool geometry number in the key-in buffer and press a search horizontal soft key. The cursor moves to an empty pot fit for the geometry.
EMPTY-SRCH.N : Searches for an empty pot forward.
EMPTY-SRCH.P : Searches for an empty pot backward.
EMPTY-SRCH : Searches for the pot nearest to the current position.

- Tool management screen

You can use bit 2 of tool information to switch between a oversize tool and normal tool. For a oversize tool, set a tool geometry number fit for the tool.

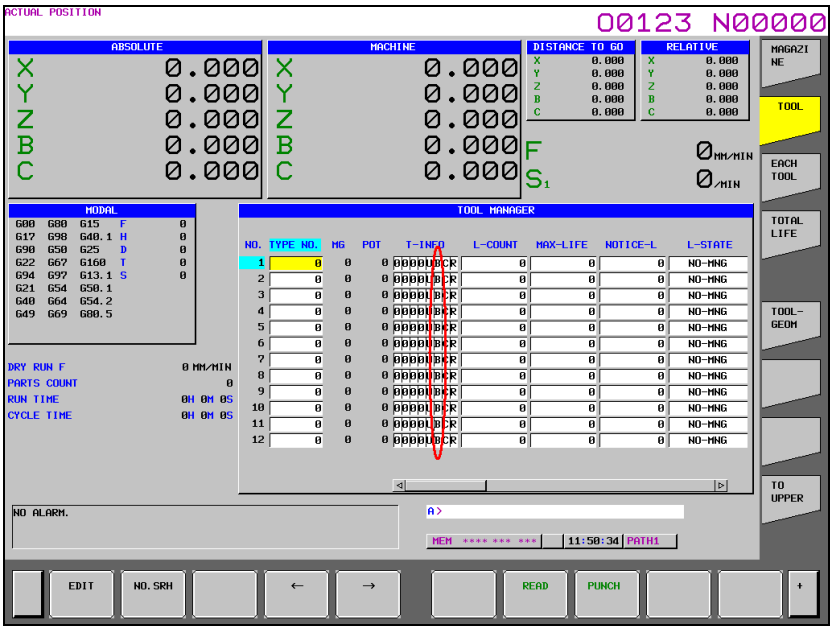


Fig. 12.3.27.5 (e) Bit for switching between a normal tool and oversize tool (15-inch display unit)

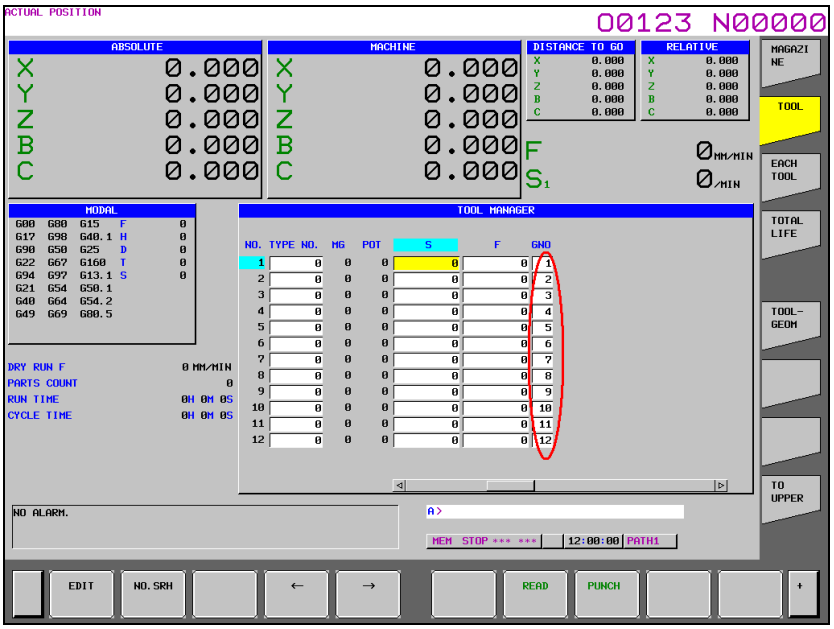


Fig. 12.3.27.5 (f) Tool geometry number (15-inch display unit)

12.3.28 Displaying and Switching the Display Language (15/19-inch Display Unit)

The language used for display can be switched to another language.
A display language can be set using a parameter. However, by modifying the setting of the display language on this screen, the display language can be switched without turning off then on the power.

Displaying and setting the display language

Procedure

1 Press function key .

2 Press vertical soft key [NEXT PAGE] several times to display soft key [LANGUAGE].

3 Press vertical soft key [LANGUAGE] to display the language screen.

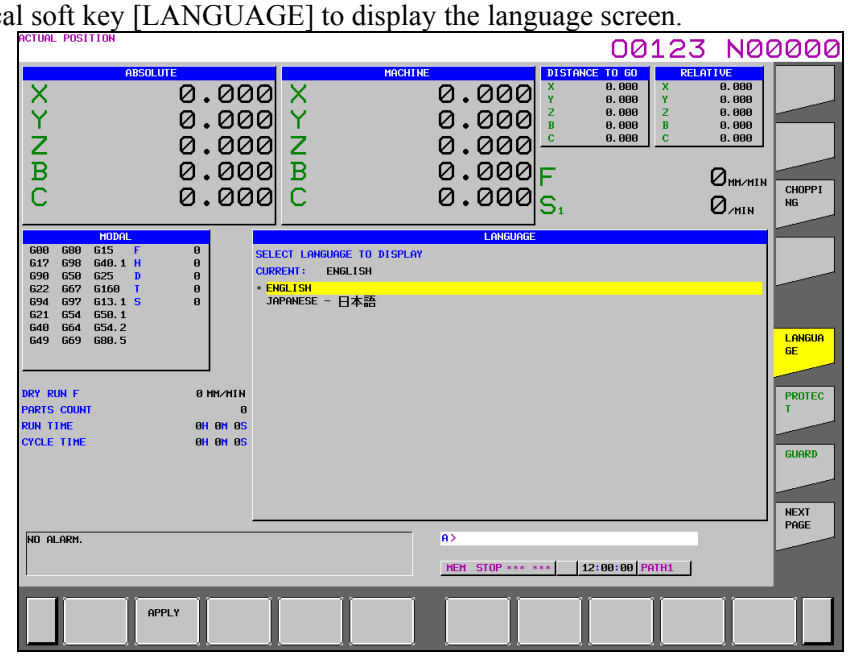


Fig. 12.3.28 (a) LANGUAGE screen (15-inch display unit)

4 Press page key  or , then press cursor key  or  to move the cursor to a desired display language.

5 Press horizontal soft key [APPLY]. The display language is switched to the selected language. The language specified on this screen continues to be used if the power is turned off then back on.

Explanation

- Language switching

The language screen can be displayed if bit 0 (NLC) of parameter No. 3280 is set to 0.

- Selectable languages

The display languages selectable on this screen are as follows:

1. English
2. Japanese
3. German
4. French
5. Chinese (Traditional)
6. Chinese (Simplified)
7. Italian
8. Korean

9. Spanish
10. Dutch
11. Danish
12. Portuguese
13. Polish
14. Hungarian
15. Swedish
16. Czech
17. Russian
18. Turkish

Among the languages listed above, English and other usable languages are displayed on the screen as a list of switchable languages.

Limitation

- Language parameter modification on the parameter screen

Which language to be used for display is specified with parameter No. 3281. This parameter can be modified using the parameter screen as well. However, if a modification is made on the parameter screen, the new setting is not reflected until "APPLY" operation is performed on the language screen or the power is turned on again. If an invalid value is set in parameter No. 3281 on the parameter screen, the screen is displayed in English after the power is turned on again.

12.3.29 Protection of Data at Eight Levels (15/19-inch Display Unit)

You can set eight CNC and PMC operation levels and one of eight protection levels for each type of CNC and PMC data.
When an attempt is made to change CNC and PMC data or output it to an external unit, the operation level is compared with the protection level to determine whether to allow the change or external output.

12.3.29.1 Operation level setting (15-inch display unit)

You can set eight CNC and PMC operation levels.

Displaying and setting the operation level setting screen

Procedure

1 Press function key .
2 Press vertical soft key [NEXT PAGE] several times to display soft key [PROTECT].
3 Press vertical soft key [PROTECT].
The operation level setting screen (Fig. 12.3.29.1 (a)) shown below is displayed.

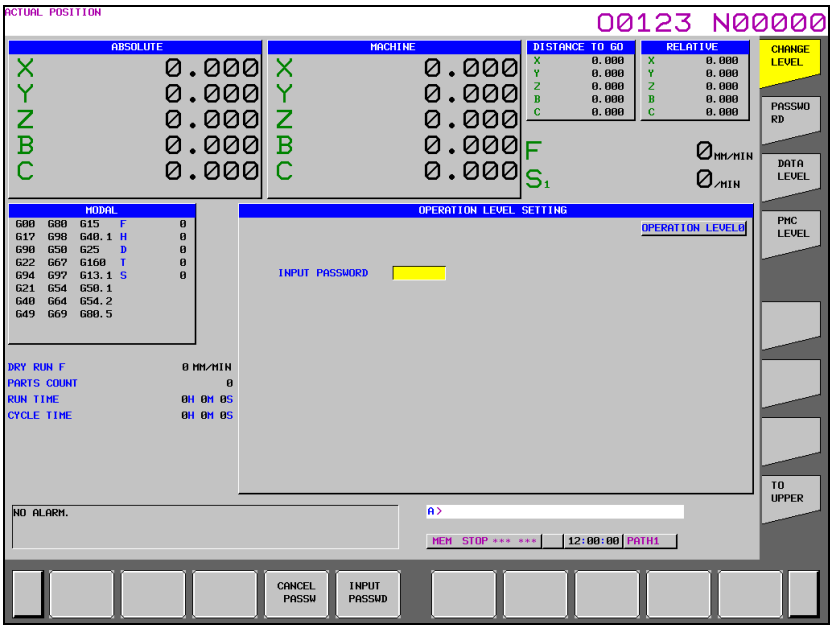


Fig. 12.3.29.1 (a) Operation level setting screen (15-inch display unit)

4 Key in the password for an operation level to be set/modified, then press horizontal soft key [INPUT PASSWD].
5 To return the operation level to 0, 1, 2, or 3, press horizontal soft key [CANCEL PASSWD].

Explanation

- Operation level setting

To select operation level 0 to 3, use the corresponding memory protection key signal.
To select operation level 4 to 7, use the corresponding password.

Table 12.3.29.1 (a) Operation level setting

Operation level	Setting	Sample grouping
7 (high)	Password	-
6	Password	MTB
5	Password	Dealer and integrator
4	Password	End user
3	Memory protection key signal	User level (level 1)
2	Memory protection key signal	User level (level 2)
1	Memory protection key signal	User level (level 3)
0 (low)	Memory protection key signal	User level (level 4)

When operation level 4 to 7 is set, the operation level remains unchanged until the password is cleared. (The operation level also remains unchanged if the power is turned off.)
Operation level 7 is reserved for CNC and PMC maintenance.

NOTE
When a password is being entered, an asterisk (*) is displayed instead of each entered character.

12.3.29.2 Password modification (15-inch display unit)

The current operation level is displayed.
The password for each of operation levels 4 to 7 can be modified.

Displaying and setting the password modification screen

Procedure

1 Press function key .
2 Press vertical soft key [NEXT PAGE] several times to display soft key [PROTECT].
3 Press vertical soft key [PROTECT].
4 Press vertical soft key [PASSWORD].
The PASSWORD CHANGE screen (Fig. 12.3.29.2 (a)) shown below is displayed.

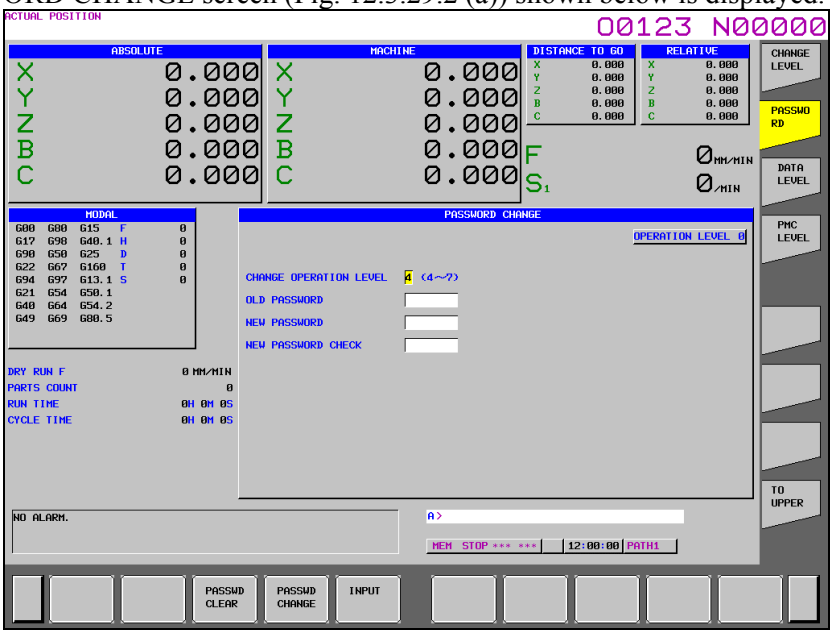


Fig. 12.3.29.2 (a) PASSWORD CHANGE screen (15-inch display unit)

5 Key in an operation level whose password is to be modified, then press horizontal soft key [INPUT].
6 Key in the current password for the operation level whose password is to be modified, then press horizontal soft key [INPUT].
7 Key in a new password, then press horizontal soft key [INPUT].
8 Key in the new password again for confirmation, then press horizontal soft key [INPUT].
9 Press horizontal soft key [PASSWD CHANGE].
10 To clear the password, press horizontal soft key [PASSWD CLEAR].

Explanation

Up to eight characters (only uppercase alphabetic characters and numeric characters) can be input.

NOTE
1 For a password, consisting of three to eight characters, the following characters are available:
- Uppercase alphabetic characters
- Numeric characters

2 When a password is being entered, an asterisk (*) is displayed instead of each entered character.

NOTE
3 Whether a password can be changed at the current operation level is determined as follows:
- Password of an operation level higher than the current operation level
  Cannot be changed.
- Password of the current operation level
  Can be changed.
- Password of an operation level lower than the current operation level
  Can be changed (only to the initial password).

4 The set password is not displayed.
Be careful not to forget the password.

12.3.29.3 Protection level setting (15-inch display unit)

The current operation level is displayed.
The change protection level and output protection level of each data item are displayed.
The change protection level and output protection level of each data item can be changed.

Confirmation based on protection level setting

Procedure

1 Press function key .
2 Press vertical soft key [NEXT PAGE] several times to display soft key [PROTECT].
3 Press vertical soft key [PROTECT].
4 Press vertical soft key [DATA LEVEL] to change the protection level of CNC data or press vertical soft key [PMC LEVEL] to change the protection level of PMC data.
The following protection level change screen (Fig. 12.3.29.3 (a)) is displayed.

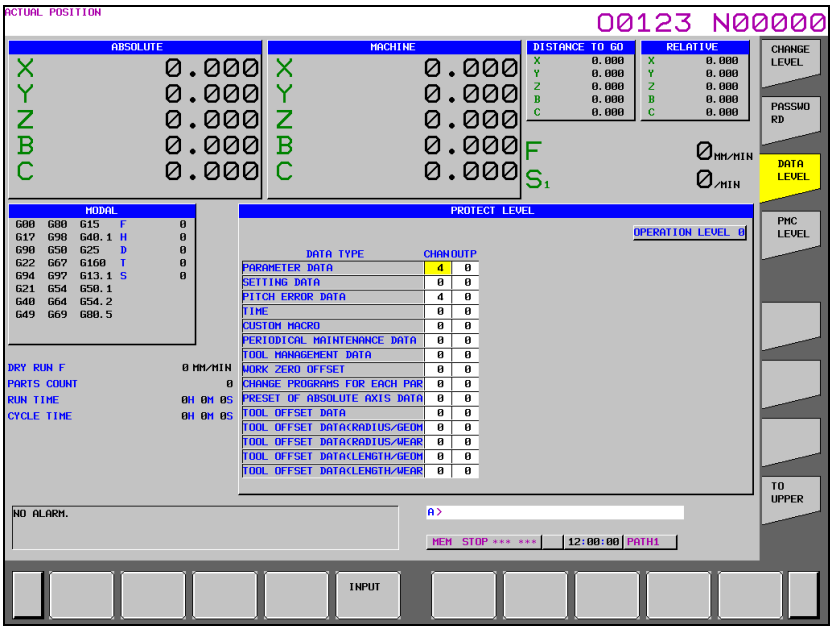


Fig. 12.3.29.3 (a) Protection level change screen (15-inch display unit)

5 Move the cursor to the change level or output level of a desired data item.
6 Key in a new desired level, then press horizontal soft key [INPUT].

NOTE
When the protection level of PMC data is set, soft key [SWITCH PMC] is used to switch between PMC paths to be set, for multi-path PMC.

Explanation

When the protection level of a data item is higher than the current operation level, the protection level of the data item cannot be changed.
The protection level of a data item cannot be changed to a protection level higher than the current operation level.
For each of the following types of data, you can set a data protection level. There are the following two types of data protection levels:

- Change protection level
  Sets the protection level used when data is changed.
- Output protection level
  Sets the protection level used when data is output to an external unit.

As a protection level, you can set a value of 0 (low) to 7 (high).

Table 12.3.29.3 (a) Protection level of each type of data

Type of data	Initial protection level	
	Change	Output
Custom macro variable data <CUSTOM MACRO> (including variable data dedicated to the macro executor)	0	0
Periodical maintenance data <PERIODICAL MAINTENANCE DATA>	0	0
Tool management data <"TOOL LIFE TIME DATA>	0	0
Tool offset data <TOOL OFFSET DATA> (For each type when tool geometry compensation and tool wear compensation are treated differently)	0	0
Clock data <TIME>	0	0
Workpiece origin shift amount data <WORK ZERO SHIFT>	0	0
Workpiece origin offset data <WORK ZERO OFFSET>	0	0
Ethernet setting data <ETHER NET SETTING>	0	0
Fixture offset data <FIXTURE OFFSET DATA>	0	0
Parameter data <PARAMETER DATA>	4	0
Settings <SETTING DATA>	0	0
Pitch error compensation data <PITCH ERROR DATA> (including 3-dimensional error compensation data)	4	0
Parameter data for Power Mate CNC manager function <PARAMETER DATA FOR POWER MATE CNC MANAGER>	0	0
Each part program <PROGRAMS FOR EACH PARTS>	0	0
Part program editing operation <CHANGE PROGRAMS FOR EACH PARTS>	0	0
Absolute coordinate preset operation <PRESET OF ABSOLUTE AXIS DATA>	0	0

Table 12.3.29.3 (a) Protection level of PMC data

Type of data	Initial protection level	
	Change	Output
Composition parameter	0	0
Setting (online)	0	0
Setting (each path)	0	0
Sequence program	0	0
PMC parameter	0	0
Timer	0	0
Counter	0	0
Keep relay	0	0
Keep relay (system)	0	0
Data table	0	0
Data table control	0	0

Type of data	Initial protection level	
	Change	Output
PMC memory	0	0
I/O configuration	0	0
I/O Link group selection	0	0
Registration of I/O device	0	0

NOTE
1 For some types of data, the output function is not provided.
2 When the protection level of data is higher than the current operation level, the protection level cannot be changed.
3 The protection level of data cannot be changed to a level higher than the current operation level.
4 Settable types of data increase or decrease, depending on the option configuration.
5 For details on the protection level of PMC data, refer to “PMC Programming Manual (B-64513EN)”.
6 Data related to tool information on the EACH TOOL DATA screen for the tool management function is protected by protecting tool offset data.
7 The type of tool offset data put in effect varies depending on the tool compensation value memory used.
8 To change the protection level for each part program, do so on the PROGRAM FOLDER screen rather than on the PROTECT LEVEL screen.
9 Part program editing includes program editing for the MDI mode.
10 Presetting absolute coordinates causes preset workpiece coordinate system values to be protected.
11 During tool offset data input/output, if any tool offset data type is not allowed to be changed or output to the outside, it is processed as follows:
- Input : Any data type other than those whose change is not allowed is changed.
- Output : Any data type other than those whose change is not allowed is output.

12.3.29.4 Setting the change protection level and output protection level of a program (15-inch display unit)

The display/operations indicated below can be performed from the directory screen.
The change protection level and output protection level of each part program are displayed.
The change protection level and output protection level of each part program can be changed.

Setting the change protection level and output protection level of a program

Procedure

1 Press function key 

.
2 Press vertical soft key [FOLDER].

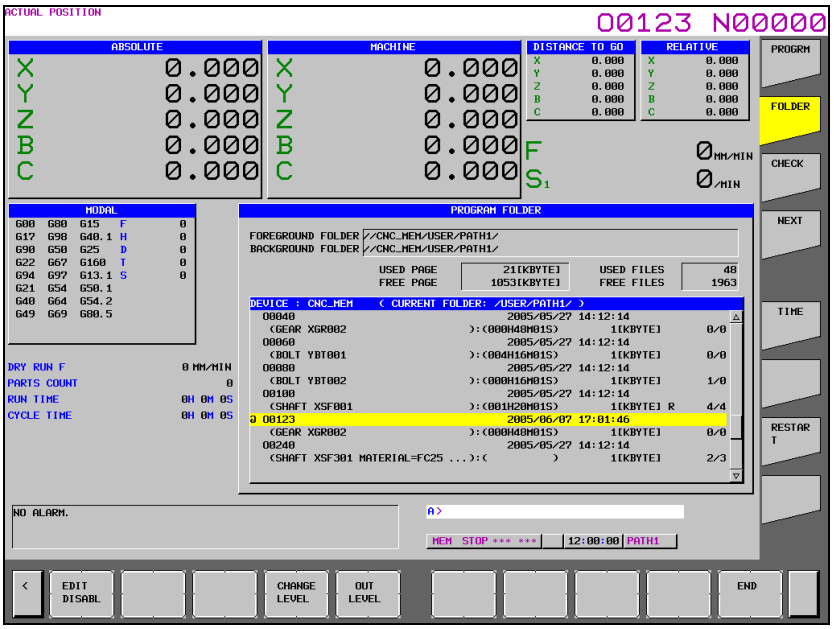


Fig. 12.3.29.4 (a) Program directory screen (15-inch display unit)

3 Press horizontal soft key [DETAIL ON] to switch to detail displays.
4 Move the cursor to a desired program.
5 Press the continuous menu key ▷ to display horizontal soft key [CHANGE ATTR].
6 Press horizontal soft key [CHANGE ATTR].
7 Key in a new desired level, then press horizontal soft key [CHANGE LEVEL].
8 To change the output protection level, key in a new desired level, then press horizontal soft key [OUT LEVEL].

Explanation

The change protection level (0 to 7) and output protection level (0 to 7) are displayed as "CHANGE PROTECTION LEVEL VALUE/OUTPUT PROTECTION LEVEL".

NOTE
1 When the protection level of data is higher than the current operation level, the protection level cannot be changed.
2 The protection level of data cannot be changed to a level higher than the current operation level.
3 A protection level can be set only for those part programs that are held on the "CNC_MEM" device.

12.3.30 Precision Level Selection (15/19-inch Display Unit)

An intermediate precision level between the parameters for emphasis on velocity (precision level 1) and the parameters for emphasis on precision (precision level 10) set on the machining parameter tuning screen (AI contour) can be selected. As shown in the Fig. 12.3.30 (a), the levels are proportionally linear, and an intermediate level can be selected so that optimal parameters can be automatically calculated to perform machining.

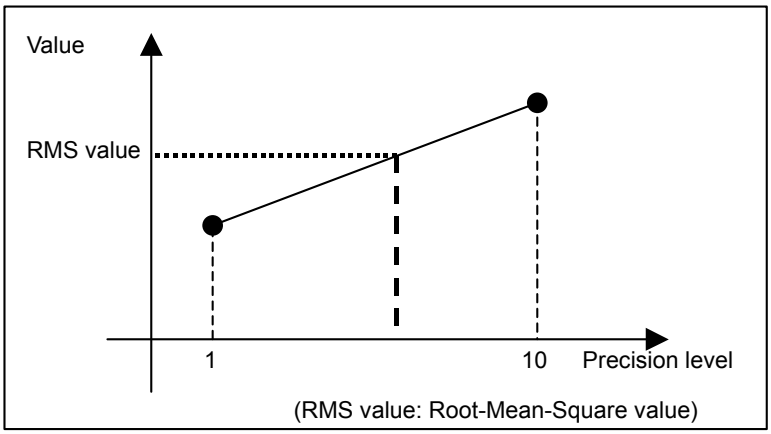


Fig. 12.3.30 (a) Image of "level"

Procedure for precision level selection

1 Select the MDI mode.

2 Press function key .

3 Press vertical soft key [NEXT PAGE] several times to display vertical soft key [PRECI LEVEL].

4 Press vertical soft key [PRECI LEVEL].

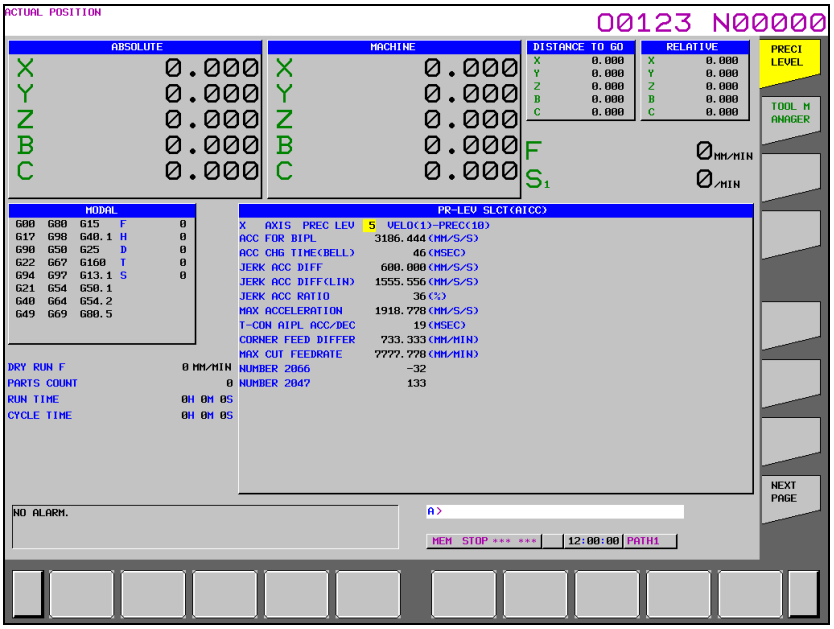


Fig. 12.3.30 (b) Precision level selection screen

5 To change the precision level, key in a desired precision level (1 to 10), then press the 

key on the MDI unit.

6 When the precision level is changed, a RMS value is obtained from the velocity-emphasized parameter set and precision-emphasized parameter set for parameter modification.
For the modified parameters, see the description of the machining parameter tuning.

7 If there is an axis in addition to the currently displayed axes, press page key 

or 

several times to display the screen for the axis.

12.3.31 Machining Level Selection (15/19-inch Display Unit)

12.3.31.1 Smoothing level selection

An intermediate smoothing level between the parameters for smoothing level 1 and the parameters for smoothing level 10 set on the machining parameter tuning screen (smoothing) can be selected. As shown in the Fig. 12.3.31.1 (a), the levels are proportionally linear, and an intermediate level can be selected so that optimal parameters can be automatically calculated to perform machining.

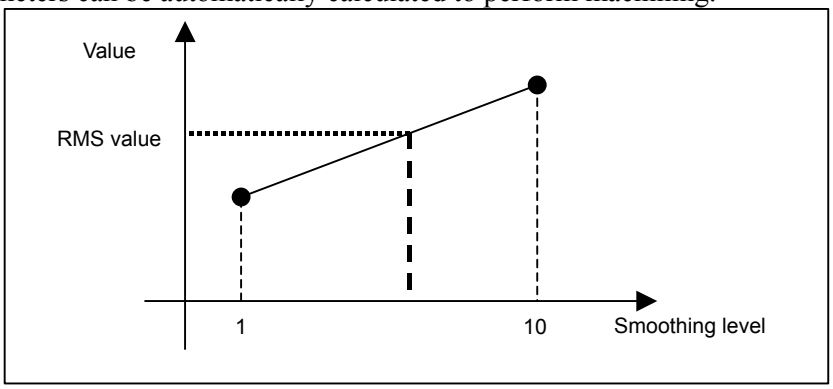


Fig. 12.3.31.1 (a) Image of "level"

Procedure for smoothing level selection

1 Select the MDI mode.

2 Press function key .

3 Press vertical soft key [NEXT PAGE] several times to display vertical soft key [MACHIN LEVEL].

4 Press vertical soft key [MACHIN LEVEL] to display vertical soft key [SMOOTH LEVEL].

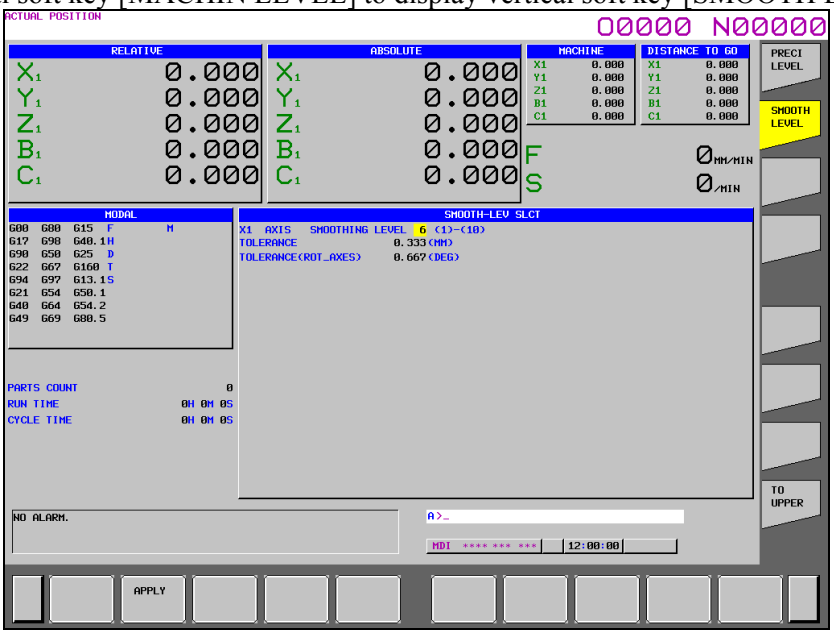


Fig. 12.3.31.1 (b) Smoothing level selection screen

5 To change the smoothing level, key in a desired smoothing level (1 to 10), then press the  key on the MDI unit.

6 When the smoothing level is changed, a RMS value is obtained from the smoothing level 1 parameter set and smoothing level 10 parameter set for parameter modification.
For the modified parameters, see the description of the machining parameter tuning.

7 If there is an axis in addition to the currently displayed axes, press page key  or  several times to display the screen for the axis.

12.3.31.2 Precision level selection

For details of precision level selection, See Subsection, “Precision Level Selection (15/19-inch Display Unit)”.

12.3.32 Machining Quality Level Selection (15/19-inch Display Unit)

Machining quality level selection allows the precision level and smoothing level to be adjusted intuitively and easily.
To display the machining quality level selection screen, set bit 6 (QLS) of parameter No. 11350 to 1.

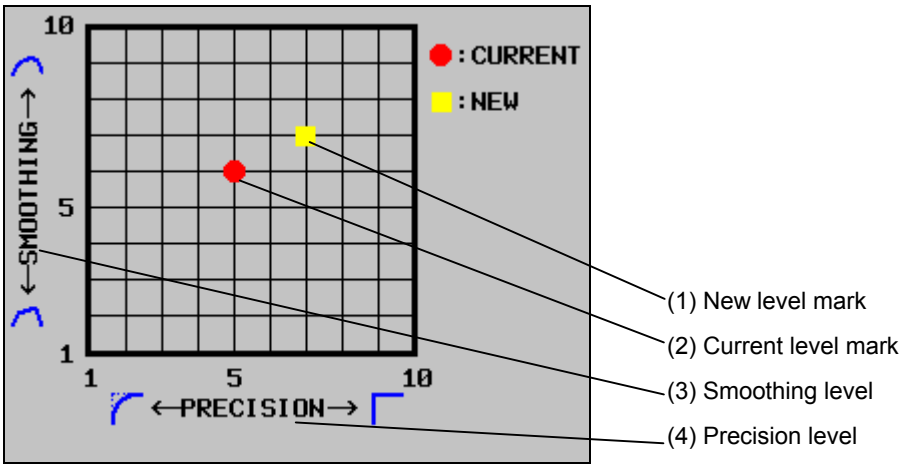


Fig. 12.3.32 (a) Quality level selection

(1) New level mark
Yellow square: Indicates the setting to be selected. (Cursor position)
(2) Current level mark
Red circle: Indicates the current setting.
(3) Smoothing level
Vertical axis: Indicates the smoothing level (1 to 10).
(4) Precision level
Horizontal axis: Indicates the precision level (1 to 10).

Procedure for machining quality level selection screen

1 Enable parameter writing.
2 Press function key .
3 Press soft key [QUALITY SELECT].

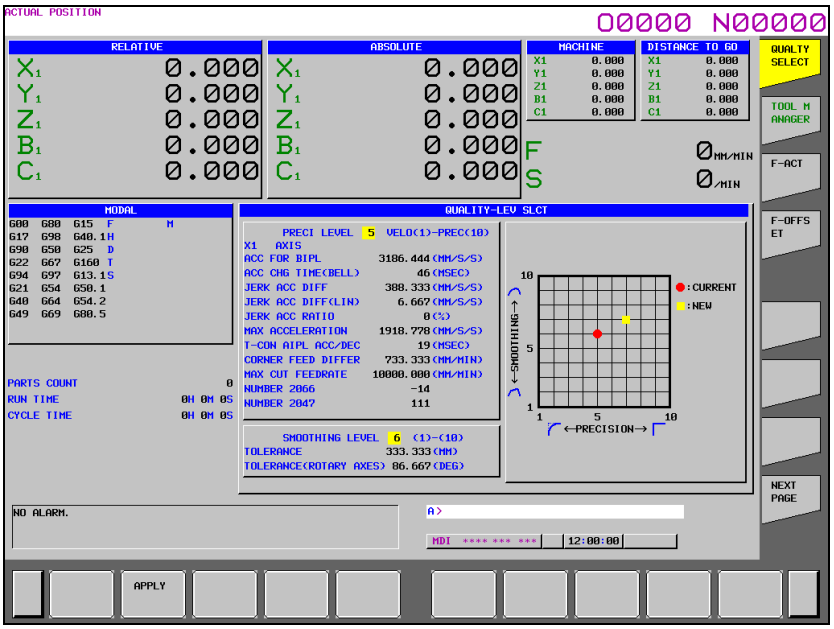


Fig. 12.3.32 (b) Machining quality level selection screen (15-inch display unit)

4 Use cursor keys to move the new level mark and select the level.
(The new level mark moves.)

5 Press soft key [APPLY] or MDI key 

to set the level.

(The current level mark moves to the position of the new level mark.)
Whether to enable or disable MDI key operation can be switched by setting the relevant parameter.

6 The set precision level and smoothing level are reflected in each setting on the PRECI LEVEL and SMOOTHING LEVEL screens displayed on the left side of the screen.

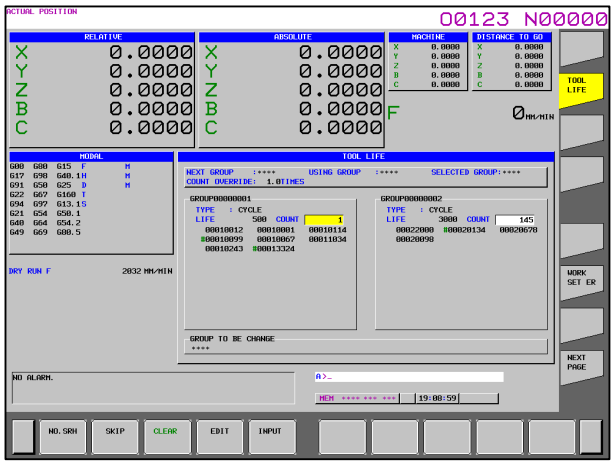
7 When the precision level or smoothing level is changed, an RMS value is obtained using the parameter settings for precision levels 1 and 10 and smoothing levels 1 and 10 and effective parameters are changed. For the changed parameters, see the description of the machining parameter tuning screen. When there is an axis other than the currently displayed axis, press a page key several times to display the screen for the desired axis.

12.3.33 Displaying and Setting Tool Life Management Data (15/19-inch Display Unit)

Displaying tool life management data on a screen enables the current status of tool life management to be grasped. Also on the screen, tool life management data can be edited. The screen is either:

- Tool life management (list screen) or
- Tool life management (group editing screen)

Overview



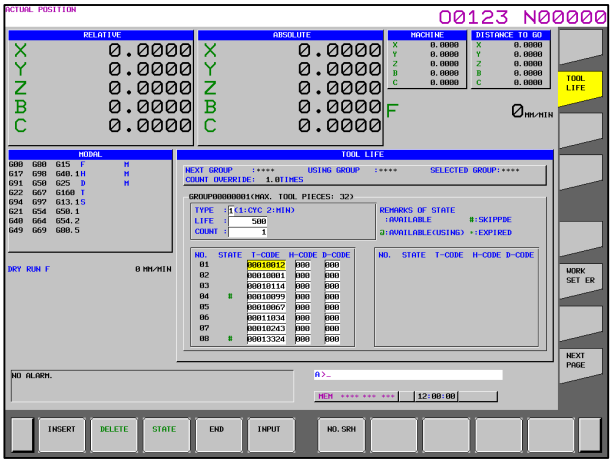
List screen

Tool life management (list screen)
Displayed items:
- NEXT GROUP
- USING GROUP
- SELECTED GROUP
- COUNT OVERRIDE
- GROUP NO.
- COUNT TYPE
- LIFE
- LIFE COUNT
- TOOL MANAGEMENT STATUS
- REGISTERED TOOL NUMBER
- GROUP TO BE CHANGE

Functions:
- Searching for groups
- Clearing execution data
- Setting the tool life counter

EDIT

END



Group editing screen

Tool life management (group editing screen)
Displayed items:
- NEXT GROUP
- USING GROUP
- SELECTED GROUP
- GROUP NO.
- MAX.TOOL PIECES
- COUNT TYPE
- LIFE
- LIFE COUNT
- T-CODE
- TOOL MANAGEMENT STATUS
- H-CODE and D-CODE
- TOOL NO.

Functions:
- Setting tool data
- Deleting group data
- Setting a tool status
- Deleting tool data
- Selecting tool skip
- Searching for groups
- Setting a life count type, life value, and tool life counter

Pressing horizontal soft key [EDIT] or [END] switches from the list screen to the group editing screen or vice versa.

M

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1), the following parameters can be used to display arbitrary groups and remaining set values.

- If bit 5 (TGN) of parameter No. 6802 = 1
  An arbitrary group is displayed on the list and group editing screens.

- If bit 3 (GRP) of parameter No. 6802 = 1
  Remaining set values are displayed on the group editing screen. The group editing screen always displays H and D codes.

T

The T series is provided with two tool change types, turret type and ATC type. For the T series, displays vary depending on which tool change type is in use.
If the turret type is in use (bit 3 (TCT) of parameter No. 5040 = 0)
- Neither the H code nor the D code is used; neither is displayed.
- No arbitrary group can be used; no arbitrary group is displayed.

If the ATC type is in use (bit 3 (TCT) of parameter No. 5040 = 1)
- The D code is displayed on the group editing screen.
- If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1) and bit 5 (TGN) of parameter No. 6802 = 1, arbitrary groups are displayed on the list and group editing screens.

If the tool life management B function is enabled (bit 4 (LFB) of parameter No.6805 = 1) and bit 3 (GRP) or parameter No. 6802 = 1, remaining set values are displayed on the group editing screen.

12.3.33.1 Tool life management (list screen) (15-inch display unit)

This screen can display the life management status of all tools in tool groups and whether the life of the tool groups has expired. It also enables you to set tool life counters and clear execution data.

Displays on the list screen

Procedure

1 Press function key .
2 Press vertical soft key [NEXT PAGE] several times to display soft key [TOOL LIFE].
3 Press vertical soft key [TOOL LIFE].

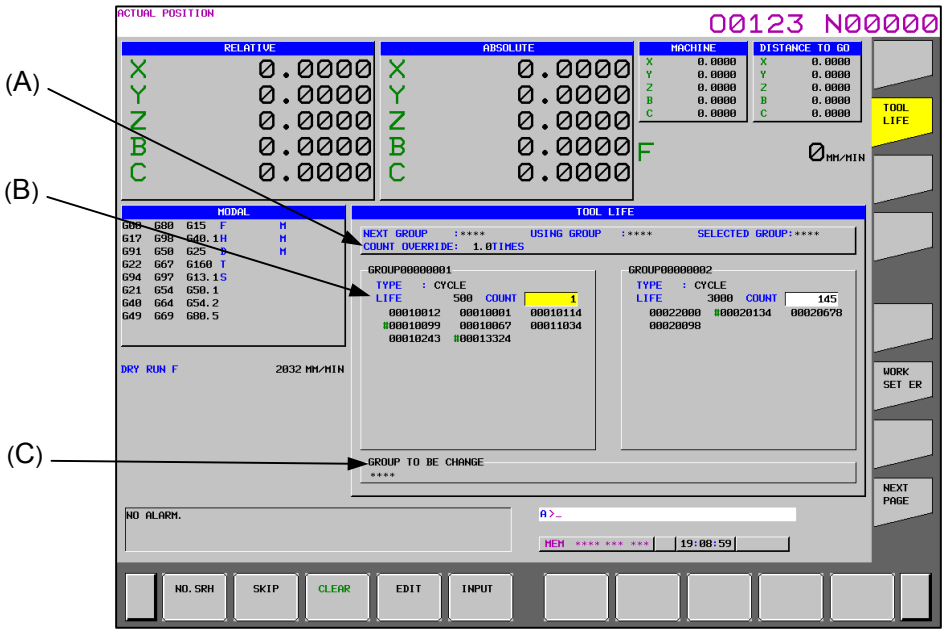


Fig. 12.3.33.1 (a) Displaying tool life management (list screen) (15-inch display unit)

- Contents of (A)

(A) displays tool group numbers and an override value. If there is no tool group to display, ****, instead of tool group numbers, is displayed.

NEXT GROUP : Tool group number for which life counting is started by the next M06 command.
USING GROUP : Tool group number for which life counting is currently under way.
SELECTED GROUP : Tool group number for which life counting is currently under way or life counting has been performed most recently.
COUNT OVERRIDE : "1.0TIMES" is displayed if the tool life counter override signal is disabled (bit 2 (LFV) of parameter No. 6801 = 0).

NOTE
If arbitrary group numbers are enabled, NEXT GROUP, USING GROUP, and SELECTED GROUP are each represented with an arbitrary group number rather than the tool group number.

- Contents of (B)

(B) displays the set life value, the current content of the tool life counter, and the registered tool numbers (in the order they are used) for each tool group. If the life count type is specification by duration, the measurement unit used in displaying and specifying set life values and tool life counter values is selected according to the setting of bit 0 (FCO) of parameter No. 6805 as listed below.

Bit 0 (FCO) of parameter No. 6805	0	1
Measurement unit used in displaying and specifying set life values and tool life counter values	1 minute	0.1 minutes

The following table lists the prefixes used with tool numbers.

Tool status	Tool in use	Tool not in use
Life remaining	@	No indication
Skip	#	#
Life expired	*	*

NOTE
1 The tool life counter indicates the count value for the tool indicated with @.
2 If bit 3 (EMD) of parameter No. 6801 = 0, a tool number remains prefixed with @ even if the life of the tool has expired until another tool is selected.
3 If bit 3 (EMD) of parameter No. 6801 =1, the following differences can occur depending on the type of the life counter in use.
- If the life counter type is specification by duration, the prefix is changed to "*" (life expired) as soon as the tool life expires.
- If the life counter type is specification by count, the counter is incremented by one at the end of a program (such as M02 or M30). So, the prefix is not changed to "*" (life expired) even if the tool life counter matches the life value. The symbol "*" (life expired) appears when the tool life count is incremented after the CNC is reset.

4 If bit 2 (ETE) of parameter No. 6804 = 1, the symbol "*" indicating the expiration of the life of the last tool appears on the tool life management screen when the life counter for the last tool of the tool group of interest matches the life value. This way, the information about the last tool in the FOCAS2 or PMC window indicates that the life of the tool has expired if the tool change signal TLCH <Fn064.0> is 1.

If arbitrary group numbers are enabled, an arbitrary group number is displayed in the parentheses beside the tool group number. If no arbitrary group number is specified, "********" is displayed instead.

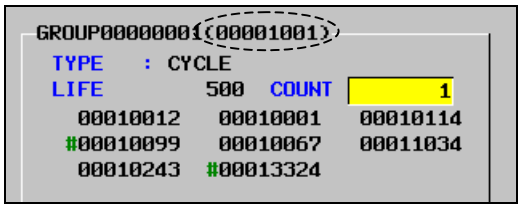


Fig. 12.3.33.1 (b) Displaying arbitrary group numbers

Arbitrary group numbers are enabled by setting the following parameters.

M

- The tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1).
- Arbitrary group numbers are enabled (bit 5 (TGN) of parameter No. 6802 = 1).

T

- The current tool change type is ATC (bit 3 (TCT) of parameter No. 5040 = 1).
- The tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1).
- Arbitrary group numbers are enabled (bit 5 (TGN) of parameter No. 6802 = 1).

- Contents of (C)

(C) displays tool group numbers for which a tool change signal has been issued.
If there are so many tool group numbers that all the numbers cannot be displayed, some are omitted, and ">>" is displayed instead.
If there is no tool group number in need of change, "****" is displayed.

NOTE
If arbitrary group numbers are enabled, a tool group number in need of change is represented with an arbitrary group number rather than the tool group number.

Setting data on the list screen

Tool life management data can be specified in the reset state (both the OP and RST signals are "0"). However, setting bit 1 (TCI) of parameter No. 6804 to 1 enables tool life management data to be specified even during automatic operation (the OP signal is "1").

NOTE
As for USING GROUP or NEXT GROUP settings:
1) During automatic operation (OP signal = "1" and bit 1 (TCI) of parameter No. 6804 = 1), only the tool life counter can be changed.
2) In the reset state (OP signal = "0" and RST signal "0"), the following editing operation stops life management because it cannot be continued.
- Clearing execution data

Procedure

- Setting the tool life counter

The tool life counter can be set with a value, using the following methods.
Method 1
1 Place the cursor on the tool life counter for a desired tool group.
2 Enter the value from the keypad.
3 Press horizontal soft key [INPUT].

Method 2

1 Place the cursor on the tool life counter for a desired tool group.
2 Enter the value from the keypad.
3 Press  key.

- Clearing execution data

All existing execution data for a tool group selected by the cursor can be cleared as follows:

1 Place the cursor on the tool group whose execution data you want to clear.
2 Press horizontal soft key [ERASE].
3 Press horizontal soft key [EXEC].

NOTE
Setting bit 4 (GRS) of parameter No. 6800 to 1 enables execution data for all registered tool groups to be cleared.

- Selecting tool groups

Tool groups can be selected using the following methods.

Method 1

1 Enter a tool group number from the keypad.
2 Press soft key [NO.SRH].

NOTE
If arbitrary group numbers are enabled, a tool group is selected by searching for an arbitrary group number rather than the tool group number.

Method 2

1 Press page key  or  to display desired groups.
2 Press cursor key  or  to move the cursor to the desired group at either the left or right.

- Switching to the group editing screen

Switch to tool life management (group editing screen).

1 Move the cursor to the tool group you want to edit:
2 Press horizontal soft key [EDIT].

12.3.33.2 Tool life management (group editing screen) (15-inch display unit)

On this screen, it is possible to edit tool life management data (such as tool life value, tool life counter, and tool data) for the tool group of interest.

Displaying the group editing screen

Procedure

1. Place the cursor on the tool group you want to edit from the list screen.
2. Press horizontal soft key [EDIT].

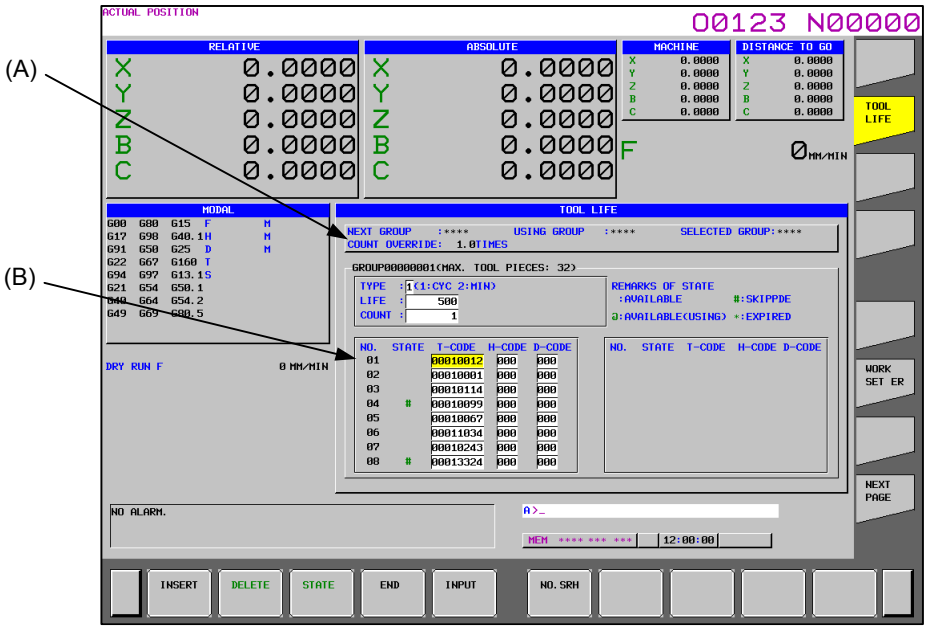


Fig. 12.3.33.2 (a) Displaying tool life management (group editing screen) (15-inch display unit)

NOTE
If no tool is registered with a tool group, none of a life count type, a life value, and a tool life counter value is displayed for the tool group.

- Contents of (A)

Like the counterpart of the list screen, (A) of the editing screen displays the next tool group number and override value. If there is no appropriate tool group, **** is displayed instead.

NEXT GROUP : Tool group for which life counting is started by the next M06 command.
USING GROUP : Tool group number for which life counting is currently under way.
SELECTED GROUP : Tool group number for which life counting is currently under way or life counting has been performed most recently.
COUNT OVERRIDE : "1.0TIMES" is displayed if the tool life counter override signal is disabled (bit 2 (LFV) of parameter No. 6801 = 0).

NOTE
If arbitrary group numbers are enabled, NEXT GROUP, USING GROUP, and SELECTED GROUP are each represented with an arbitrary group number rather than the tool group number.

- Contents of (B)

(B) details tool life management data related to a selected tool group as follows:

TYPE :

1	Specification by count
2	Specification by duration

LIFE : Tool life value
COUNT : Tool life counter

STATE :

Tool status	Tool in use	Tool not in use
Life remaining	@	No indication
Skip	#	#
Life expired	*	*

T-CODE : Tool number

M

H-CODE : Tool length compensation specification code
D-CODE : Cutter compensation specification code

T

H-CODE : No display.
D-CODE : Tool offset value specification code
if ATC type is in use (bit 3 (TCT) of parameter No. 5040 = 1)

NOTE

1 The tool life counter indicates the count value for the tool indicated with @.
2 If bit 3 (EMD) of parameter No. 6801 = 0, a tool number remains prefixed with @ even if the life of the tool has expired until another tool is selected.
3 If bit 3 (EMD) of parameter No. 6801 = 1, the following differences can occur depending on the type of the life counter in use.
- If the life counter type is specification by duration, the prefix is changed to "*" (life expired) as soon as the tool life expires.
- If the life counter type is specification by count, the counter is incremented by one at the end of a program (such as M02 or M30). So, the prefix is not changed to "*" (life expired) even if the tool life counter matches the life value. The symbol "*" (life expired) appears when the tool life count is incremented after the CNC is reset.

4 If bit 2 (ETE) of parameter No. 6804 = 1, the symbol "*" indicating the expiration of the life of the last tool appears on the tool life management screen when the life counter for the last tool of the tool group of interest matches the life value. This way, the information about the last tool in the FOCAS2 or PMC window indicates that the life of the tool has expired if the tool change signal TLCH <Fn064.0> is 1.

Displaying arbitrary groups and remaining set values

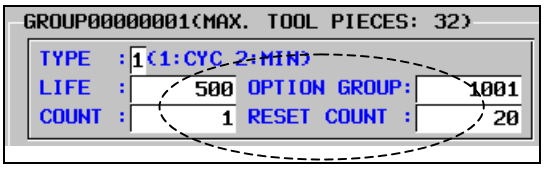


Fig. 12.3.33.2 (b) Displaying arbitrary group numbers and remaining set values on the editing screen

M

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1) and the following parameters are enabled, arbitrary groups and remaining set values are displayed.
OPTION GROUP : Arbitrary group number (if bit 5 (TGN) of parameter No. 6802 = 1)
REST COUNT : Remaining set value used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1)

T

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1)
Turret type (bit 3 (TCT) of parameter No. 5040 = 0)
OPTION GROUP : No display.

REST COUNT : Remaining set value used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1)

ATC type (bit 3 (TCT) of parameter No. 5040 = 1)
OPTION GROUP : Arbitrary group number (if bit 5 (TGN) of parameter No. 6802 = 1)
REST COUNT : Remaining set value used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1)

Setting data on the group editing screen

Tool life management data can be specified in the reset state (both the OP and RST signals are "0"). However, setting bit 1 (TCI) of parameter No. 6804 to "1" enables tool life management data to be specified even during automatic operation (the OP signal is "1").
The available editing operations are listed below.

M

Items that can be edited	Mode
Setting a life count type, life value, tool life counter, and tool data (T code, H code, and D code)	All modes
Adding tool numbers (T code)	MDI
Deleting all tool group data at a time	MDI
Deleting tool data (status, T code, H code, and D code)	MDI
Selecting tool skip	MDI
Specifying to clear tool data (life re-set)	MDI

T

Items that can be edited	Mode
Setting a life count type, life value, tool life counter, and tool data (T code and D code) However, the D code can be edited only if the ATC type is in use (bit 3 (TCT) of parameter No. 5040 = 1).	All modes
Adding tool numbers (T code)	MDI
Deleting all tool group data at a time	MDI
Deleting tool data (status, T code, and D code)	MDI
Selecting tool skip	MDI
Specifying to clear tool data (life re-set)	MDI

M
T

If no tool is registered with a tool group, none of a life count type, a tool life value, and a tool life counter value can be set for the tool group. First of all, add a tool number (T code).

NOTE
1 As for USING GROUP or NEXT GROUP editing:
<1> During automatic operation (OP signal = "1" and bit 1 (TCI) of parameter No. 6804 = 1), only the tool life counter can be changed.
<2> In the reset state (OP signal = "0" and RST signal "0"), the following editing operation stops life management because it cannot be continued.
- Adding tool numbers (T code)
- Deleting all tool group data at a time
- Deleting tool data (status, T code, H code, and D code)

NOTE
2 The following editing operations may set the tool change signal to “1”.
- Selecting tool skip for the last tool.
- Deleting tool numbers, resulting in any tool other than those whose life has expired or who have been skipped being not found in the tool group of interest.

3 The following editing operations may reset the tool change signal to “0”.
- Adding tool numbers, leading to tools whose life has not expired being set in the tool group of interest.
- Selecting tool clear.

Procedure

- Setting a life count type, tool life value, tool life counter, tool data, arbitrary group number, and remaining set value

Setting a life count type, tool life value, tool life counter, tool data, arbitrary group number, and remaining set value

Method 1
1 Place the cursor on the desired item.
2 Enter a value from the keypad.
3 Press horizontal soft key [INPUT].

Method 2
1 Place the cursor on the desired item.
2 Enter a value from the keypad.
3 Press 

.

NOTE
1 Changing a tool life value or tool life counter does not affect the tool status or tool change signal.
2 Changing a life count type causes the tool life value and too life counter to be reset to 0.

M

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1) and the following parameters are enabled, arbitrary groups and remaining set values can be specified
OPTION GROUP : Arbitrary group number (if bit 5 (TGN) of parameter No. 6802 = 1)
REST COUNT : Remaining set value used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1)

T

If the tool life management B function is enabled (bit 4 (LFB) of parameter No. 6805 = 1)
Turret type (bit 3 (TCT) of parameter No. 5040 = 0)
OPTION GROUP : Cannot be set.
REST COUNT : It is possible to set a remaining set value to be used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1).
ATC type (bit 3 (TCT) of parameter No. 5040 = 1)
OPTION GROUP : Arbitrary group numbers can be set (if bit 5 (TGN) of parameter No. 6802 = 1).
REST COUNT : It is possible to set a remaining set value to be used until a new tool is selected (if bit 3 (GRP) of parameter No. 6802 = 1).

- Adding tool numbers

Tool numbers can be added to a tool group as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) just before a tool number to be added.
3 Enter the tool number from the key pad.
4 Press horizontal soft key [INSERT].

(Example)

Adding tool number 1550 between numbers 1 and 2 (for the M series)

1 Move the cursor to the data for number 1, enter “1550”, and press [INSERT].

NO.	STATE	T-CODE	H-CODE	D-CODE
01		00010012	000	000
02		00010001	000	000
03		00010114	000	000

2 The entered T code 1550 is inserted in the position of number 2. The H and D codes are reset to 0.

NO.	STATE	T-CODE	H-CODE	D-CODE
01		00010012	000	000
02		00001550	000	000
03		00010001	000	000

- Deleting all tool group data at a time

All tool group data can be deleted at a time as follows:

1 Select the MDI mode.
2 Select the tool group from which you want to delete all tool data at a time.
3 Press horizontal soft key [DELETE].
4 Press horizontal soft key [GROUP].
5 Press horizontal soft key [EXEC].

- Deleting tool data

Tool data can be deleted from a tool group as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) you want to delete.
3 Press horizontal soft key [DELETE].
4 Press horizontal soft key [<CURS>].

NOTE
1 Deleting all tools from a tool group is equivalent to deleting the tool group itself.
2 Deleting a tool indicated with @ (in use) results in @ being moved to the previous tool whose life has expired or which has been skipped.

- Selecting tool skip

Tool data can be placed in a skip state as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) for a tool you want to skip.
3 Press horizontal soft key [STATE].
4 Press horizontal soft key [SKIP].

- Specifying to clear tool data (life re-set)

Tool data state can be cleared as follows:

1 Select the MDI mode.
2 Place the cursor on the tool data (T code, H code, or D code) for a tool you want to clear.
3 Press horizontal soft key [STATE].

4 Press horizontal soft key [CLEAR].

- Selecting a tool group

A tool group can be selected as follows:
Method 1
1 Enter a tool group number from the keypad.
2 Press horizontal soft key [NO.SRH].

NOTE
If arbitrary group numbers are enabled, a tool group is selected by searching for an arbitrary group number rather than the tool group number.

Method 2
1 Press page key  or  to display the target tool group.

- Switching to the list screen

The tool life management (list screen) can be resumed as follows:
1 Press horizontal soft key [END].

12.3.34 Displaying and Setting Workpiece Setting Error Compensation Data (15/19-inch Display Unit)

An amount of error used in workpiece setting error compensation can be set on the workpiece setting error screen.
The workpiece setting error screen is displayed as follows:

1 Press function key .

2 Press vertical soft key [WORK SET ER].
The workpiece setting error screen appears.

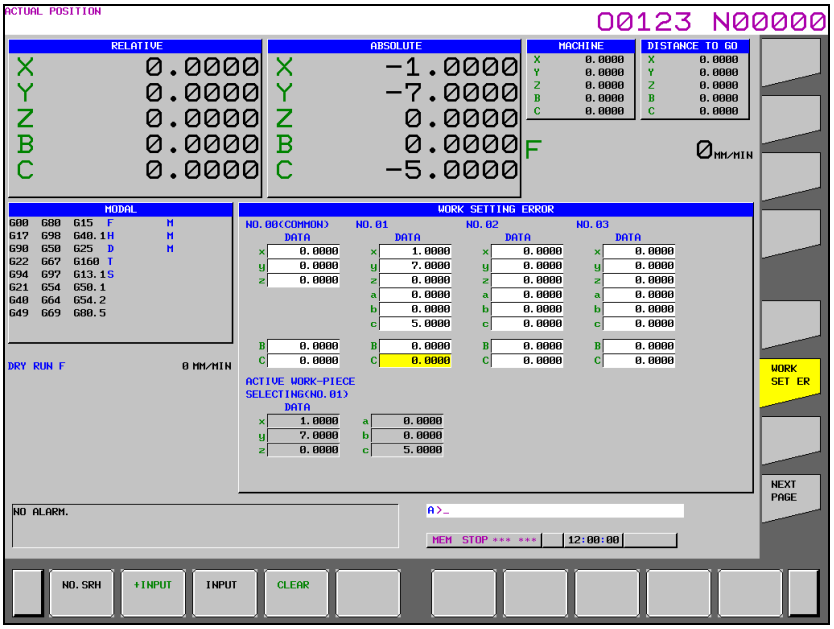
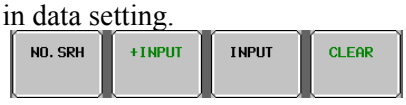


Fig. 12.3.34 (a) Workpiece setting error screen (15-inch display unit)

Up to 7 sets (No. 01 to No. 07) of different amounts of workpiece setting error can be specified. In addition, it is possible to specify an amount of error (No. 00 (COMMON)) to be added to an error in the 7 sets in common.
The X-direction error Δx, Y-direction error Δy, and Z-direction error Δz are specified, respectively, in the x, y, and z fields. The rotational errors Δa, Δb, and Δc are specified, respectively, in the a, b, and c fields.
Table rotation axis positions 1 and 2 are specified in the fields under the a, b, and c fields (C and B in the figure).
Rotation axis names specified in parameters Nos. 19681 and 19686 are displayed in the axis name fields.
Table rotation axis positions 1 and 2 are specified for rotation axes. No position must be specified for axes (including virtual axes) other than table rotation axes. So, no such item is displayed for setting.
Table rotation axis position setting items are displayed under the following conditions.

- It must have been correctly specified which axis is a table rotation axis.
  If parameter No. 19680 = 12 (table rotation type), axis numbers must have been specified correctly in parameters Nos. 19681 and 19686.
  If parameter No. 19680 = 21 (mixed type), an axis number must have been specified correctly in parameter No. 19686.
- Bit 0 of parameter No. 1006 must be 1 (rotation axis) for the axes specified above.

The following soft key can be used in data setting.



- After a workpiece setting error number you want to display is entered, pressing horizontal soft key [NO.SRH] causes a setting screen for the target workpiece setting error to appear.
- After a number is entered, pressing horizontal soft key [+INPUT] causes the number to be added.
- After a number is entered, pressing horizontal soft key [INPUT] causes the number to be set.
- Pressing horizontal soft key [CLEAR] causes the next soft key to appear.



- After a workpiece setting error number to be deleted is entered, pressing horizontal soft key [NO.] deletes the data corresponding to the workpiece setting error number.
- Pressing horizontal soft key [ALL] deletes data for all workpiece setting error numbers.

12.3.35 Displaying and Setting Pattern Data Inputs (15/19-inch Display Unit)

Described below are a method for displaying machining menus (pattern menus) created by machine tool builders and a method for setting them. The descriptions are based on an example. For actual pattern menus and pattern data, refer to manuals released by respective machine tool builders.

Displaying pattern data and pattern menus

Given below is a procedure for displaying pattern menus.

1 Press function key .
2 Press vertical soft key [NEXT PAGE] several times.
3 Press vertical soft key [PATTERN MENU].
The pattern menu screen (Fig. 12.3.35 (a)) shown below appears.

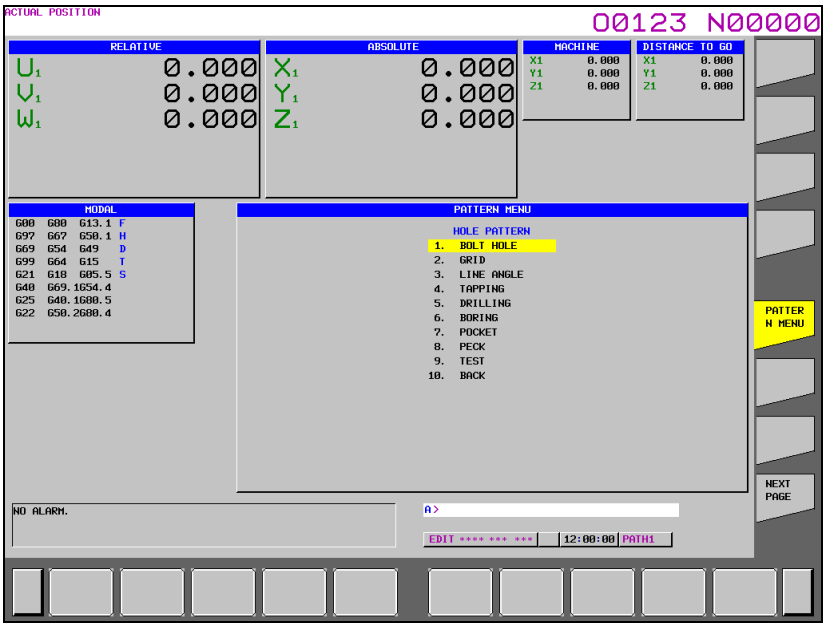


Fig. 12.3.35 (a) Pattern menu screen (15-inch display unit)

On this screen, a pattern to be used can be selected.
The following two methods can be used to select patterns.

- Using the cursor

  1 Move the cursor to a pattern name you want to select, using cursor key ↑ or ↓, and then press soft key [SELECT] or [INPUT].

- Specifying a pattern number

  1 Enter a number displayed at the left side of a pattern name, and press soft key [SELECT] or [INPUT].

  The custom macro screen (pattern data screen) (Fig. 12.3.35 (b)) shown below appears.

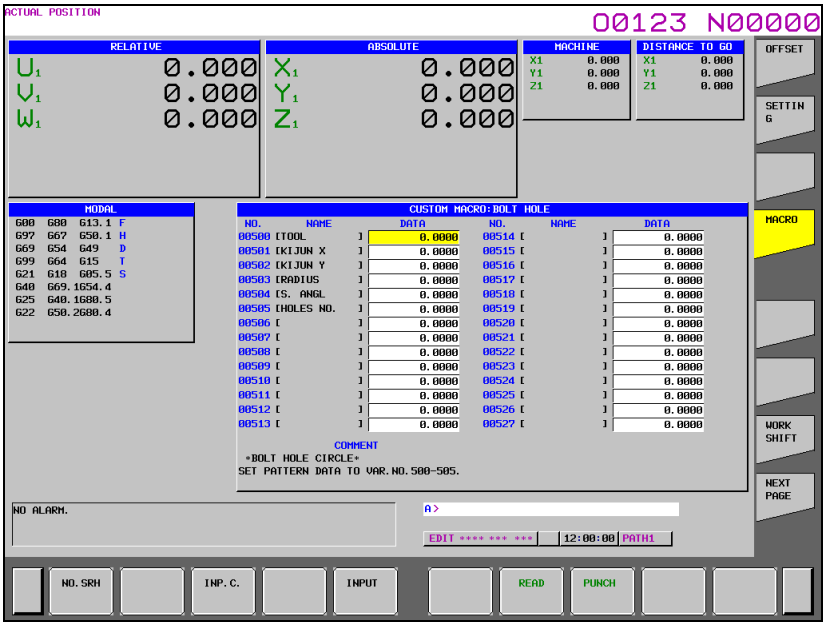


Fig. 12.3.35 (b) Custom macro screen (pattern data) (15-inch display unit)

  2 Enter the necessary pattern data, and press [INPUT].

  3 After entering all necessary data, select the MEM mode and press the cycle start button. Machining begins.

Explanation

- Explanations about the pattern menu screen

HOLE PATTERN

An arbitrary character string consisting of 12 or less characters can be displayed as a menu title.

BOLT HOLE

An arbitrary character string consisting of 10 or less characters can be displayed as a pattern name.

Machine tool builders should program menu title and pattern name character strings, using custom macros, and save them in program memory.

- Explanations about the custom macro screen (pattern data screen)

BOLT HOLE

An arbitrary character string consisting of 12 or less characters can be displayed as a pattern data title.

TOOL.

An arbitrary character string consisting of 10 or less characters can be displayed as a variable name.

BOLT HOLE CIRCLE
The comment text consists of nine 12-character blocks, or the comment can be up to 12 lines (10.4-inch display unit) or 8 lines (8.4-inch display unit) with one block counted as one line.

Machine tool builders should program variable name and comment text character strings, using custom macros, and save them in program memory.

12.3.36 Built-in 3D Interference Check

On the setting screens of built-in 3D interference check function, the following operations can be performed:

- Sets each target figure for interference check.
- Checks the current setting.

The screens are arranged in a hierarchical structure as shown in Fig.12.3.36 (a)

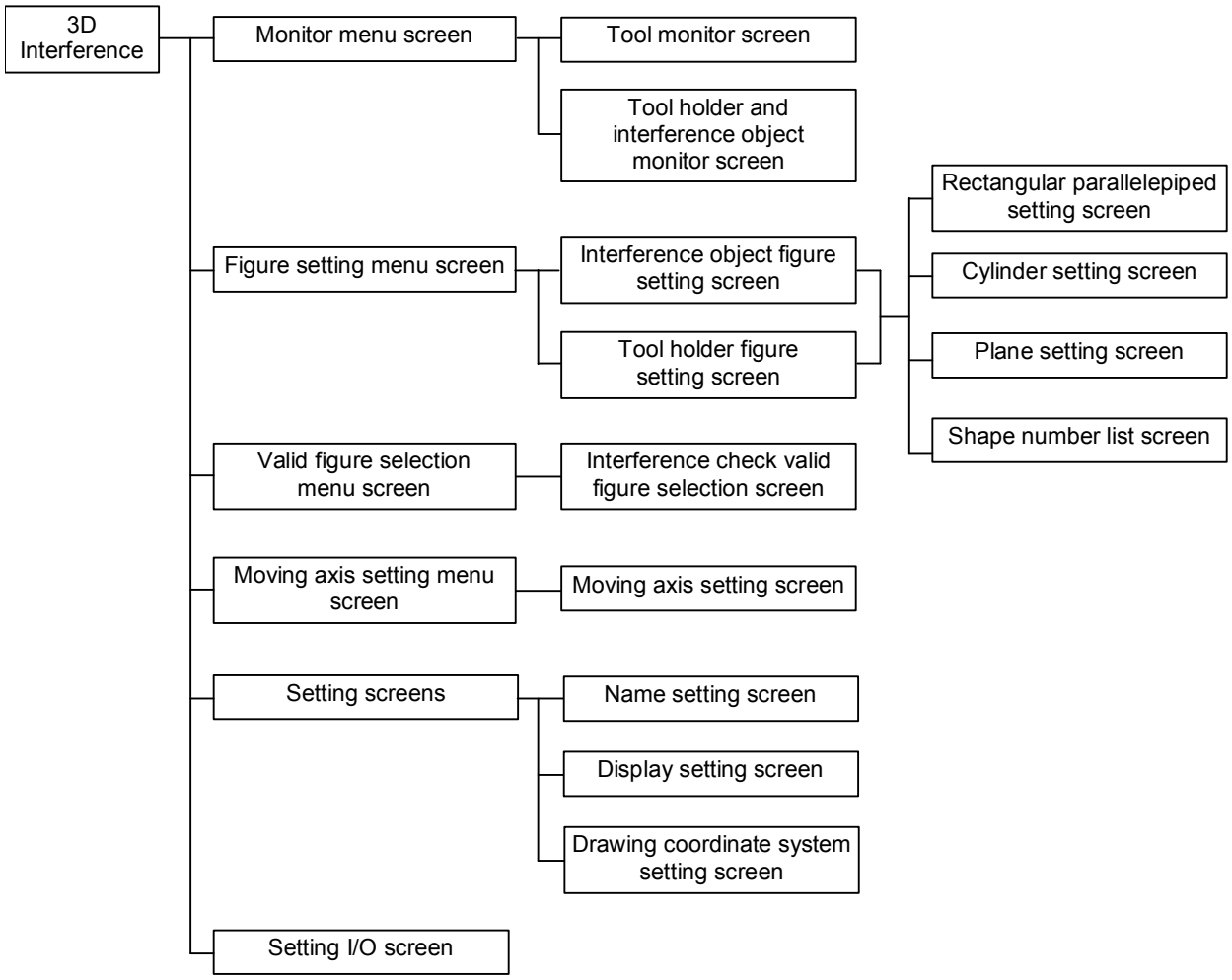


Fig.12.3.36 (a)

12.3.36.1 Monitor menu screen

Screen configuration

On the monitor menu screen, select a tool, tool holder, or object to display information related to the selected item.
The following information is displayed according to the type of target item:
TOOL : Tool number and offset number
TOOL HOLDER, OBJECT : Figure number

In a multi-path system, not a monitor screen for each path, but a common monitor screen is displayed.

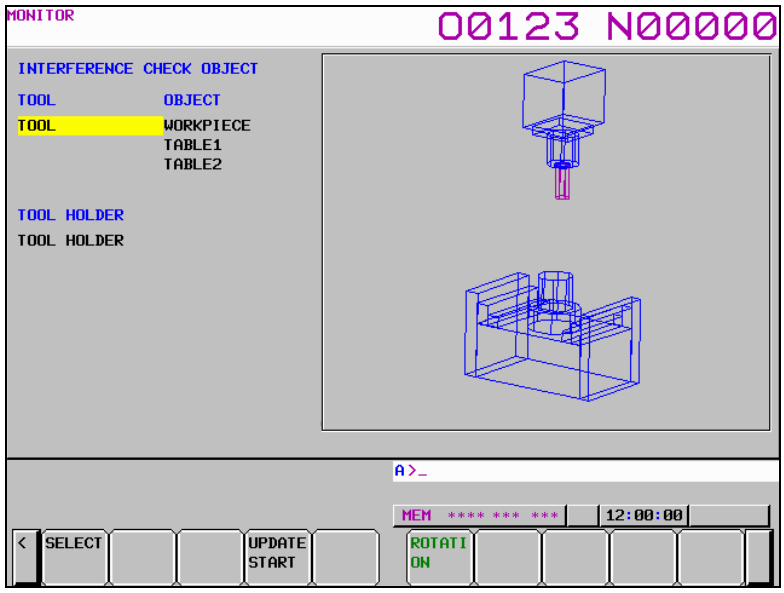


Fig. 12.3.36.1 (a) Monitor menu screen 10.4”

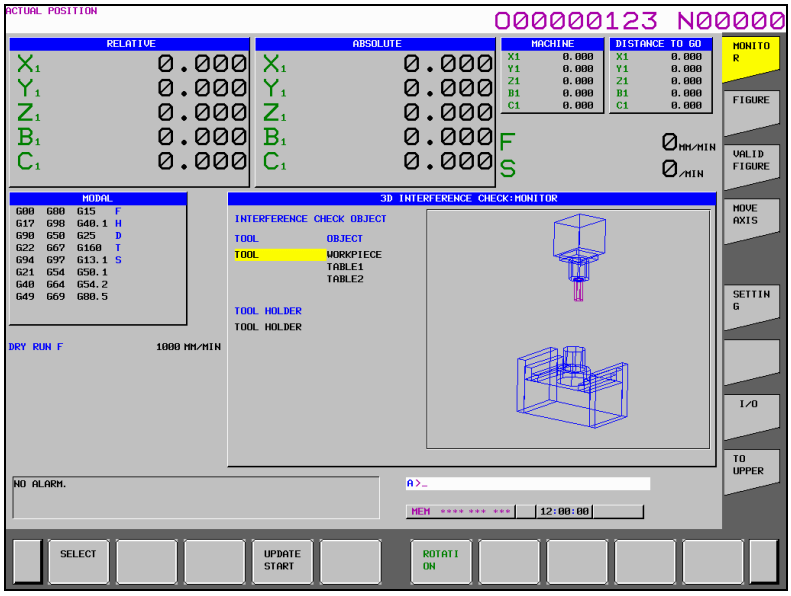


Fig. 12.3.36.1 (b) Monitor menu screen 15”

To display the monitor menu screen, use the following procedure:

1. Press function key .
2. Press soft key [3D INTER.].
3. Press soft key [MONITOR].

NOTE
Set the names of tools, tool holders, and objects which are to interfere with a tool on the relevant setting screen described later.

Operation

On the monitor menu screen, the following soft keys are available to perform operation:

[SELECT] Selects a target item. Performs the same operation as MDI key .

[UPDATE START] Updates the drawing of the figure for check at regular intervals.

[UPDATE STOP] Stops updating the drawing of the figure for check.
[ROTATION] Rotates the figure for check.

Soft key [UPDATE START] or [UPDATE STOP] is displayed when it can be used depending on the drawing update state.

Pressing soft key [ROTATION] causes soft keys for rotation operation to appear.

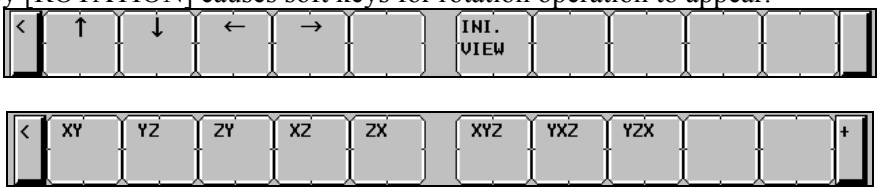


Fig. 12.3.36.1 (c) Soft keys for rotation operation

Soft key [↑] Rotates the figure for check upward.
Soft key [↓] Rotates the figure for check downward.
Soft key [←] Rotates the figure for check to left.
Soft key [→] Rotates the figure for check to right.
Soft key [INI. VIEW] Moves the figure for check to the initial position.
Soft key [XY] Displays the figure for check in drawing coordinate system XY.
Soft key [YZ] Displays the figure for check in drawing coordinate system YZ.
Soft key [ZY] Displays the figure for check in drawing coordinate system ZY.
Soft key [XZ] Displays the figure for check in drawing coordinate system XZ.
Soft key [ZX] Displays the figure for check in drawing coordinate system ZX.
Soft key [XYZ] Displays the figure for check in drawing coordinate system XYZ.
Soft key [YXZ] Displays the figure for check in drawing coordinate system YXZ.
Soft key [YZX] Displays the figure for check in drawing coordinate system YZX.

Procedure for changing the drawing coordinate system

The drawing coordinate system can be changed to change the display in the specified coordinate system at a time.
To change the drawing coordinate system, use the following procedure:

1 Press soft key [ROTATION].
2 Press soft key [XY], [YZ], [ZY], [XZ], [ZX], [XYZ], [YXZ], or [YZX] corresponding to the drawing coordinate system to which you want to change.
The figure for check is displayed in the selected coordinate system and the soft keys shown in Fig. 12.3.36.1 (d) appear.

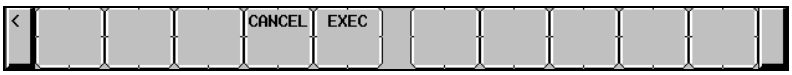


Fig. 12.3.36.1 (d) Soft keys displayed when the drawing coordinate system is changed

3 Press soft key [EXEC]. Pressing soft key [EXEC] determines the change of the drawing coordinate system. To cancel the change of the drawing coordinate system, press soft key [CANCEL]. The figure for check returns to the state present before coordinate system selection.

NOTE
After the restart of the CNC, the drawing coordinate system changed by rotation operation returns to the drawing coordinate system set on the drawing coordinate system setting screen.

12.3.36.2 Tool monitor screen

Screen configuration

When a tool is selected on the monitor menu screen, the tool number and offset number are displayed.
When bit 2 (ICT) of parameter No. 10930 is set to 0, the tool number is not displayed.

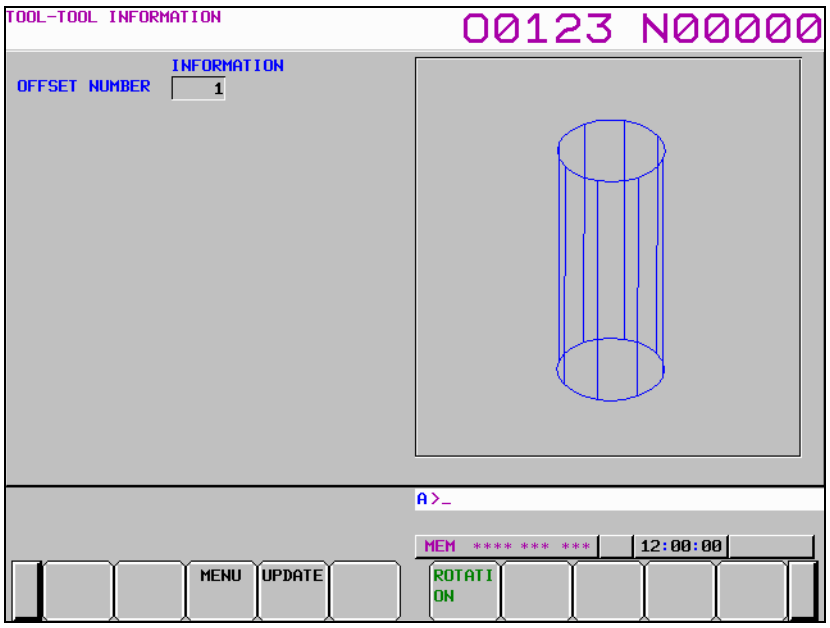


Fig. 12.3.36.2 (a) Tool monitor screen 10.4”

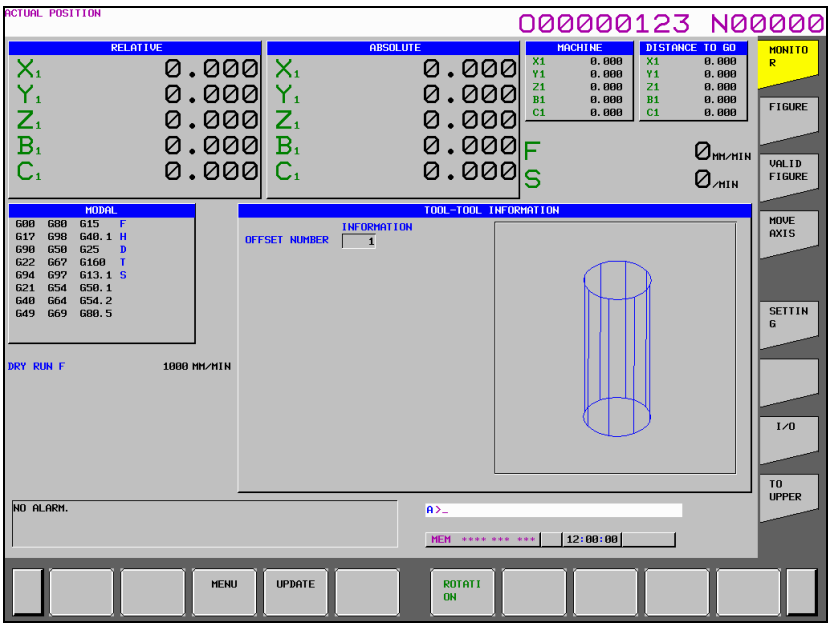


Fig. 12.3.36.2 (b) Tool monitor screen 15”

Operation

On the tool monitor screen, the following soft keys are available to perform operation:

[MENU] Displays the monitor menu screen.
[UPDATE] Updates the display of the figure for check.
[ROTATION] Rotates the figure for check.

Pressing soft key [ROTATION] causes soft keys for rotation operation to appear.



Fig. 12.3.36.2 (c) Soft keys for rotation operation

Soft key [↑] Rotates the figure for check upward.
Soft key [↓] Rotates the figure for check downward.
Soft key [←] Rotates the figure for check to left.
Soft key [→] Rotates the figure for check to right.

12.3.36.3 Tool holder and object monitor screen

Screen configuration

When a tool holder or an object which is to interfere with a tool is selected on the monitor menu screen, the figure number is displayed.

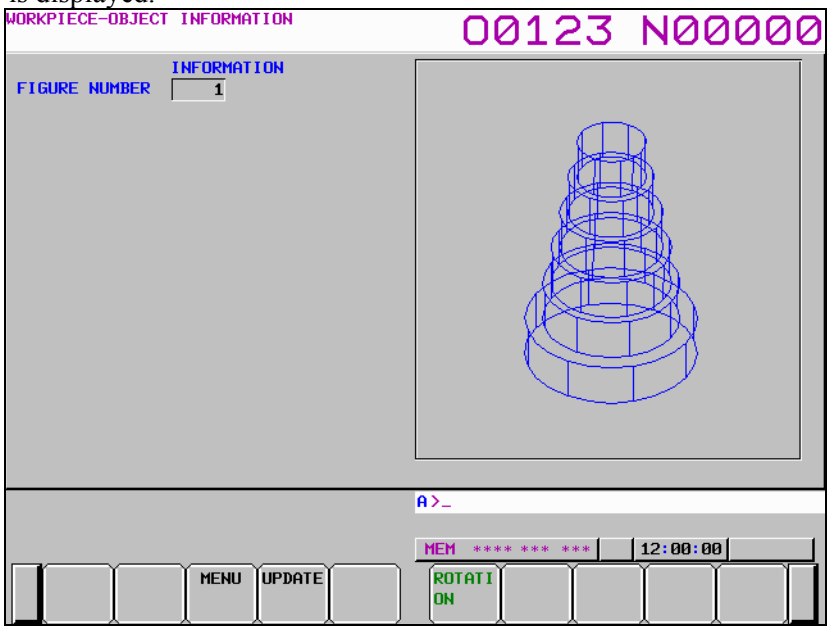


Fig. 12.3.36.3 (a) Object monitor screen 10.4”

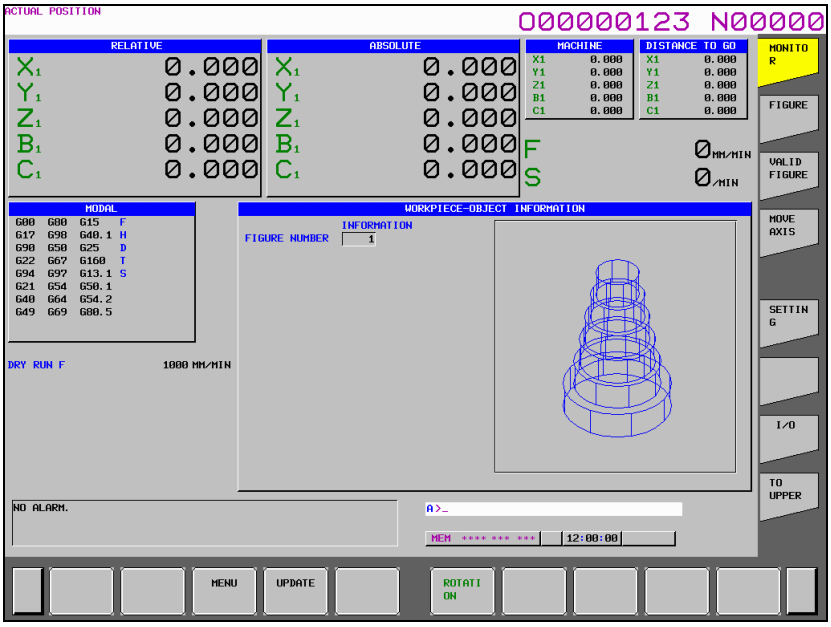


Fig. 12.3.36.3 (b) Object monitor screen 15”

Operation

On the tool holder and object monitor screen, the following soft keys are available to perform operation:

[MENU] Displays the monitor menu screen.
[UPDATE] Updates the display of the figure for check.
[ROTATION] Rotates the figure for check.

Pressing soft key [ROTATION] causes soft keys for rotation operation to appear.



Fig. 12.3.36.3 (c) Soft keys for rotation operation

Soft key [↑] Rotates the figure for check upward.
Soft key [↓] Rotates the figure for check downward.
Soft key [←] Rotates the figure for check to left.
Soft key [→] Rotates the figure for check to right.

12.3.36.4 Figure setting menu screen

On the figure setting menu screen, select an object or tool holder.
All target items are displayed. The item at the cursor is highlighted.

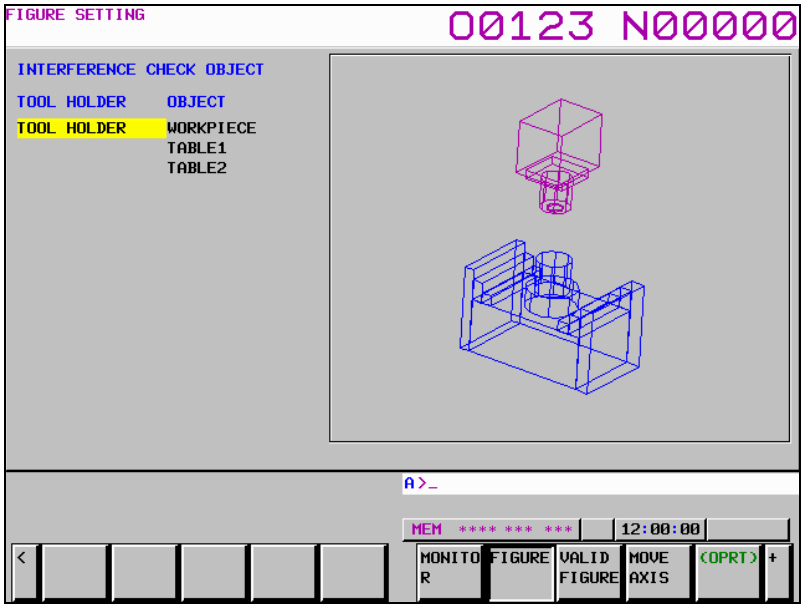


Fig. 12.3.36.4 (a) Figure setting menu screen 10.4”

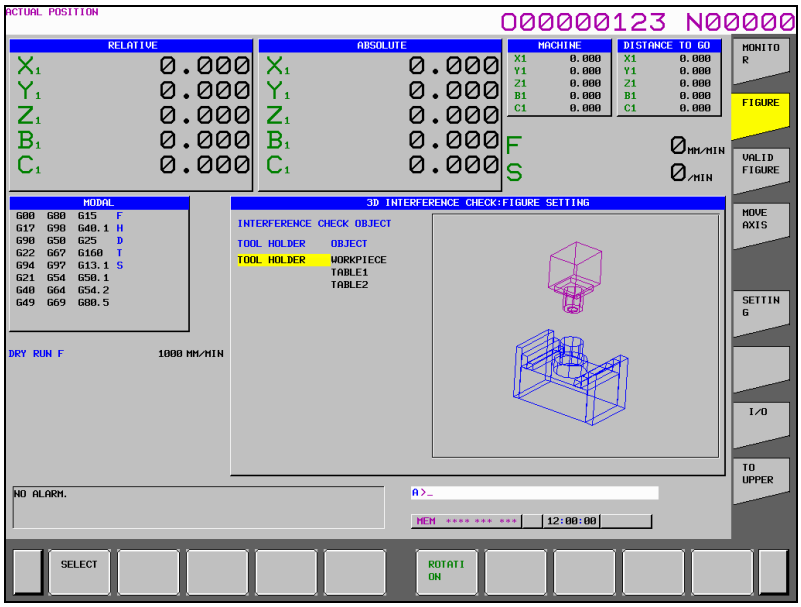


Fig. 12.3.36.4 (b) Figure setting menu screen 15”

The maximum number of objects that can be set is 3 in 1-path control and parameter ENO (No.10930#6) is set to 0. The maximum number of objects that can be set is 6 in 1-path control and parameter ENO (No.10930#6) is set to 1 or multi-path control.
The maximum number of tool holders that can be set is 1 in 1-path control or 4 in multi-path control.

To display the figure setting menu screen, use the following procedure:

1. Press function key .
2. Press soft key [3D INTERF.].
3. Press soft key [FIGURE].

NOTE
Set the “names of objects” and “names of tool holders” on the relevant setting screen described later.

Select an object or tool holder on the menu screen and press soft key [SELECT] or the <INPUT> key to display the relevant figure setting screen.

In a multi-path system, not data for each path, but data common to the system is set.
Data may be set for any path.
The following explanations of screens are common to objects 1 to 6 and tool holders 1 to 4.

12.3.36.5 Object figure setting screen

Screen configuration

When an object is selected on the menu screen, the screen shown in Fig. 12.3.36.5 (a) or Fig. 12.3.36.5 (b) appears.
Specify each shape that makes up the figure of an object to define the figure of the object. To define the figure of an object, set the following data items:

- Figure element 1, 2, 3, 4, 5, 6:
  Specify each figure element that makes up the figure of an object. Up to six figure elements can be input. In the SHAPE NO. field, specify a value between 1 and the number of shapes (number of displayed shapes set on the display setting screen). When you want to specify no shape, specify 0.

- Reference position:
  Reference position used when a figure is defined
- Reference angle 1(master rotation axis):
  Angle of the master rotation axis to be assumed for the measurement of each figure when the object rotates as the rotation axis is moved
- Reference angle 2(slave rotation axis):
  Angle of the slave rotation axis to be assumed for the measurement of each figure when the object rotates as the rotation axis is moved
- Numerical unit: Machine unit or input unit

Up to 10 figures can be defined for each object. The setting screen can be changed for a desired figure using page keys or [FIGURE NO.SRH].

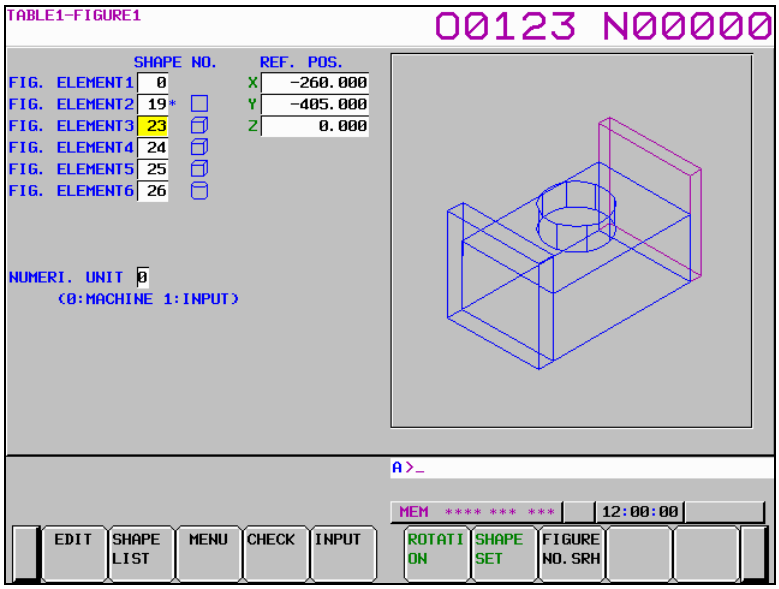


Fig. 12.3.36.5 (a) Object figure setting screen 10.4”

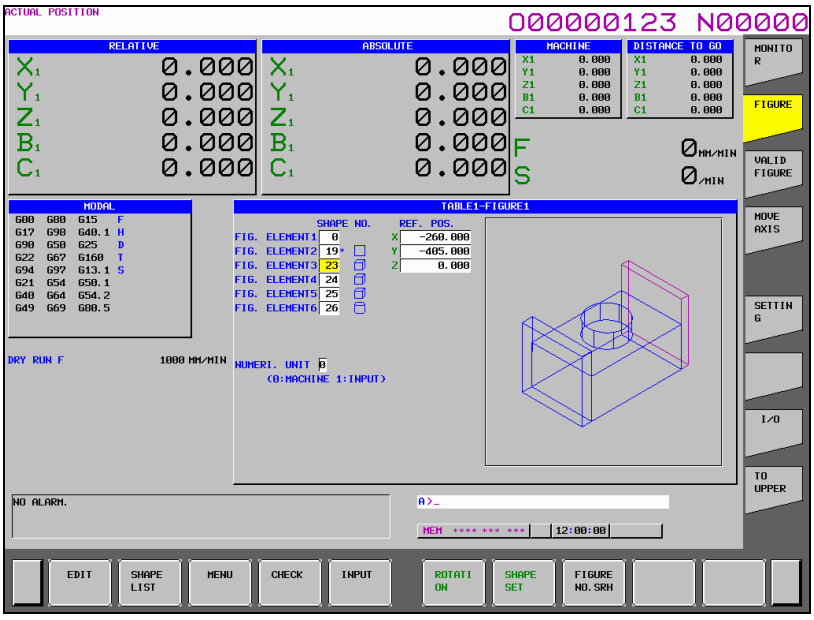


Fig. 12.3.36.5 (b) Object figure setting screen 15”

In the above setting example, figure 1 consists of five rectangular parallelepipeds (Figure element 2 = Shape number 19, Figure element 3 = Shape number 23, Figure element 4 = Shape number 24, Figure element 5 = Shape number 25, and Figure element 6 = Shape number 26).

If no shape is defined for a figure, alarm PS0492, "3DCHK FIG. ILLEGAL: [String1]" is issued.
If data is not set or is invalid for a set shape number, an asterisk (*) is displayed to the right of the shape number.
Move the cursor to Figure element 1 to Figure element 6 and press soft key [EDIT]. The setting screen for the specified shape number appears.

> ⚠ **CAUTION**
> The setting you make is not updated until you turn the power off, then on again, or set built-in 3D interference check setting change signal TDICHG to "1".

Shape type display

To the right of each shape number, the icon indicating the shape type selected as the figure element is displayed.
Each icon indicates a shape type as follows:

: Rectangular parallelepiped
: Cylinder
: Plane

When shape data is not set, no shape type icon is displayed.

Operation

On the object figure setting screen, the following soft keys are available to perform operation:

[EDIT]	Edits shape data for the shape number selected by the cursor.
[SHAPE LIST]	Displays the shape number list screen.
[MENU]	Displays the figure setting menu screen.
[CHECK]	Updates the display for checking the input shape.
[INPUT]	Inputs data. Performs the same operation as MDI key INPUT.
[ROTATION]	Rotates the figure for check.
[SHAPE SET]	Sets a new shape or changes the shape type.
[FIGURE NO.SRH]	Displays the figure setting screen for the input figure number.

Pressing soft key [ROTATION] causes soft keys for rotation operation to appear.



Fig. 12.3.36.5 (c) Soft keys for rotation operation

Soft key [↑] Rotates the figure for check upward.
Soft key [↓] Rotates the figure for check downward.
Soft key [←] Rotates the figure for check to left.
Soft key [→] Rotates the figure for check to right.

Procedure for setting new shape data

To set new shape data, use the following procedure:

1. Move the cursor to the SHAPE NO. field for a figure element you want to set.
2. Enter a shape number and press the INPUT key or soft key [INPUT].
3. Press soft key [SHAPE SET]. Pressing soft key [SHAPE SET] causes the following soft keys to appear:



Fig. 12.3.36.5 (d) Shape type soft keys

4. Press the soft key corresponding to the shape type you want to set. Pressing the soft key selects the relevant shape type and displays the relevant setting screen.

Procedure for changing the shape type

To change the shape type of set shape data, use the following procedure:

1. Move the cursor to the SHAPE NO. field for the figure element of which shape type you want to change.
2. Press soft key [SHAPE SET]. Pressing soft key [SHAPE SET] causes the following soft keys to appear:
   At this time, the soft key corresponding to the currently set shape type does not appear.

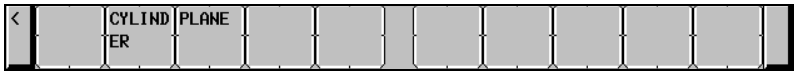


Fig. 12.3.36.5 (e) Soft keys displayed when the current setting is rectangular parallelepiped



Fig. 12.3.36.5 (f) Soft keys displayed when the current setting is cylinder



Fig. 12.3.36.5 (g) Soft keys displayed when the current setting is plane

3. Press the soft key corresponding to the shape type you want to set. Pressing the soft key causes a confirmation message and the following soft keys to appear:



Fig. 12.3.36.5 (h) Soft keys displayed after a shape is selected

4. Pressing soft key [EXEC] causes the setting screen for the selected shape type to appear.
   The shape data before change is cleared.
   To cancel the change of the shape type, press soft key [CANCEL].

Procedure for editing shape data

To edit shape data, use the following procedure:

1. Move the cursor to the SHAPE NO. field for a figure element you want to edit.

2. Press soft key [EDIT]. Pressing soft key [EDIT] causes the setting screen for the set shape type to appear. When no shape type is set, the rectangular parallelepiped setting screen appears.

12.3.36.6 Tool holder figure setting screen

Screen configuration

When a tool holder is selected on the menu screen, the following screen appears.
Specify each shape that makes up the figure of a tool holder to define the figure of the tool holder. To define the figure of a tool holder, set the following data items:

- Figure element 1, 2, 3, 4, 5, 6

Specify each figure element that makes up the figure of a tool holder. Up to six figure elements can be input. In the SHAPE NO. field, specify a value between 1 and the number of shapes (number of displayed shapes set on the display setting screen). When you want to specify no shape, specify 0.

NOTE
Please make the figure which the tip is attached to as figure element 1 of the tool holder. The tip figure is put on figure element 1 of the tool holder by automatically making of the tool.
The following tool is attaching the tip.
- General-purpose tool, threading tool, grooving tool, round-nose tool, point-nose straight tool and multifunction tool

- Reference position: Reference direction used when a shape is defined
- Tool reference position: Position of the tool reference position (XYZ) viewed from the reference position
- Tool tip direction: Tool tip direction viewed from the tool reference position

NOTE
Tool tip direction is set for the following tool.
- Drill, counter sink tool, flat end mill, ball end mill, tap, reamer, boring tool, face mill, side tool for thread, back tool.

- Reference angle 1(master rotation axis):
  Angle of the master rotation axis to be assumed for the measurement of each figure when the tool holder rotates as the rotation axis is moved
- Reference angle 2(slave rotation axis):
  Angle of the slave rotation axis to be assumed for the measurement of each figure when the tool holder rotates as the rotation axis is moved
- Numerical unit: Machine unit or input unit

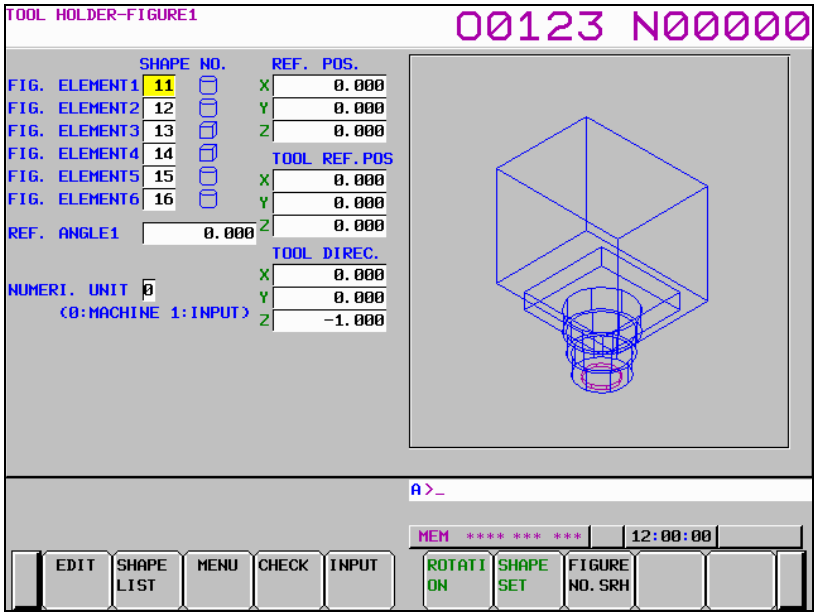


Fig. 12.3.36.6 (a) Tool holder figure setting screen 10.4”

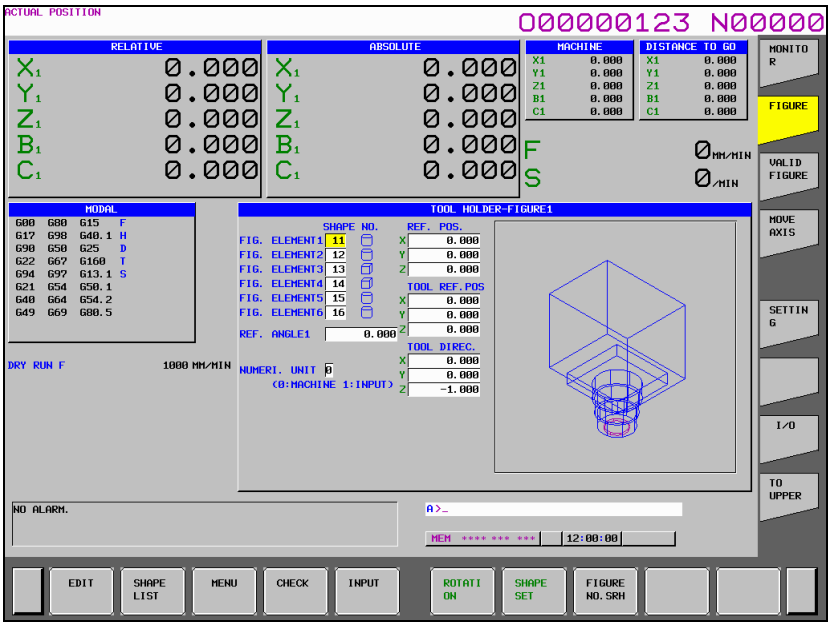


Fig. 12.3.36.6 (b) Tool holder figure setting screen 15”

The number of figures that can be defined for tool holders is as follows:

Up to 120 in a 1-path system
Up to 60 in a 2-path system
Up to 40 in a 3-path system
Up to 30 in a system having 4 or more paths
（For a system having 4 or more paths, the allowable number of tool holders is 4.）

The setting screen can be changed for a desired figure using page keys or [FIGURE NO.SRH].
Use the “the tool geometry size data” to specify the tools associated with each tool holder figure. When a tool is selected, a tool holder figure is automatically selected. You can check the currently selected figure on the monitor screen.
If no shape is defined for a figure, alarm PS0492, “3DCHK FIG. ILLEGAL: [Target name]” is issued.
If data is not set or is invalid for a set shape number, an asterisk (*) is displayed to the right of the shape number.
Move the cursor to Figure element 1 to Figure element 6 and press soft key [EDIT]. The setting screen for the specified shape number appears.

> ⚠ **CAUTION**
> The setting you make is not updated until you turn the power off, then on again, or set built-in 3D interference check setting change signal TDICHG to “1”.

Shape to be automatically made as tool figure

Kind of shape to be automatically made as tool figure is decided according to Table12.3.36.6 (a).

Table12.3.36.6 (a)

Tool type	Making method	Kind of shape to be automatically made
General-purpose tool	Making method 1	Rectangular parallelepiped
Threading tool	Making method 1	Rectangular parallelepiped
Grooving tool	Making method 1	Rectangular parallelepiped
Round-nose tool	Making method 1	Rectangular parallelepiped
Point nose straight tool	Making method 1	Rectangular parallelepiped
Multifunctional tool	Making method 1	Rectangular parallelepiped
Drill	Making method 2	Cylinder
Counter sink tool	Making method 2	Cylinder
Flat end mill	Making method 2	Cylinder

Tool type	Making method	Kind of shape to be automatically made
Ball end mill	Making method 2	Cylinder
Tap	Making method 2	Cylinder
Reamer	Making method 2	Cylinder
Boring tool	Making method 2	Cylinder
Face mill	Making method 2	Cylinder
Angle tool	Making method 2	Cylinder
Side tool for drill	Making method 2	Cylinder
Probe	-	No figure
Side tool for thread	Making method 3	Three rectangular parallelepiped or two cylinder and one rectangular parallelepiped (Note)
Back tool	Making method 4	Four rectangular parallelepiped or three cylinder and one rectangular parallelepiped (Note)

NOTE
The making method is different according to the kind of figure element 1 of the tool holder, which is rectangular parallelepiped or cylinder.

Making method 1

The tip figure of the following tool is put on figure element 1 of the tool holder by automatically making of the tool. Please make the figure which the tip is attached to as figure element 1 of the tool holder.

- General-purpose tool
- Threading tool
- Grooving tool
- Round-nose tool
- Point-nose straight tool
- Multifunction tool

The tip figure is made by the following method based on element 1 of the tool holder figure.

- Thickness of the tip (a) : The opposite direction of adjacent vertex 1 of figure element 1
- Length of the tip (b) : The same direction of adjacent vertex 2 of figure element 1
- Width of the tip (c) : The same direction of adjacent vertex 3 of figure element 1

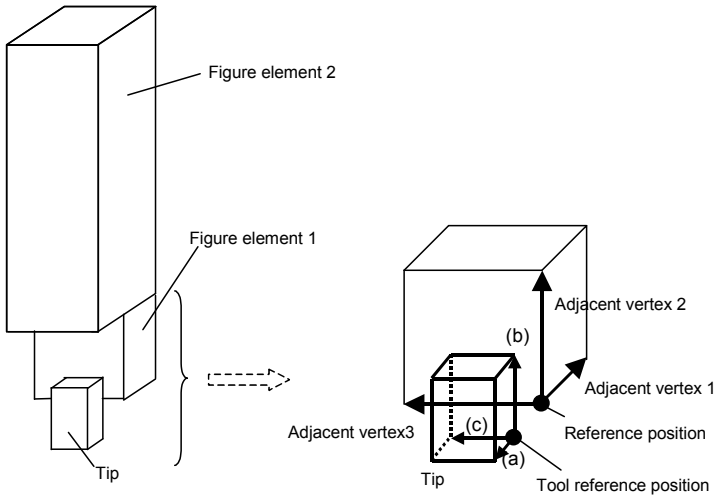


Fig.12.3.36.6 (c)

When the parameter TTD (No.10959#0) is set to 1, the method of making the tip figure is changed by tool setting of the tool geometry size data. Different tip figure can be defined at the same tool reference position by changing tool setting.

The method of making tip figure corresponding to each tool setting is as follows.

1. When tool setting of the each tool is set as Table12.3.36.6 (b), tip figure is made as Fig.12.3.36.6 (d).

Table12.3.36.6 (b)

Tool type	Tool setting
General-purpose tool	1 2 3 4 9 10 11 12
Threading tool	1 2 3 4 9 10 11 12
Grooving tool	1 2 3 4 9 10 11 12
Round-nose tool	1 2 3 4 9 10 11 12
Point nose straight tool	5 6 7 8
Multifunctional tool	1 2 3 4 9 10 11 12

- Thickness of the tip (a) : The opposite direction of adjacent vertex 1 of figure element 1
- Length of the tip (b) : The same direction of adjacent vertex 2 of figure element 1
- Width of the tip (c) : The same direction of adjacent vertex 3 of figure element 1

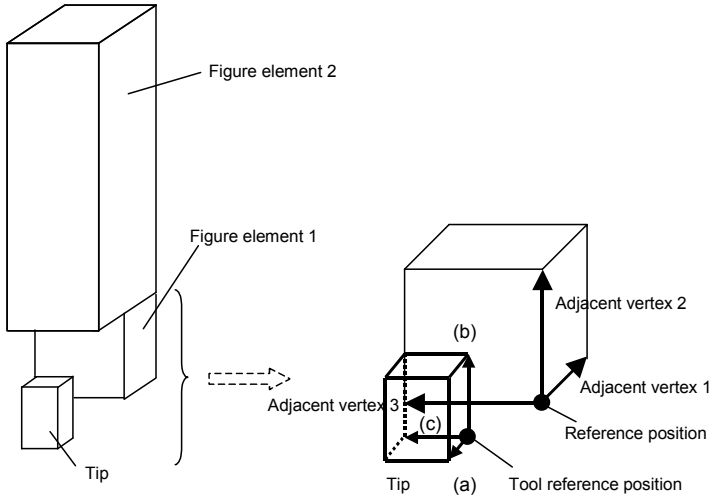
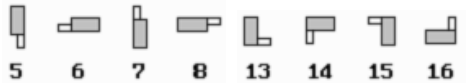
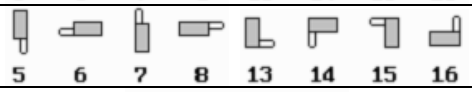
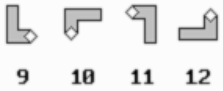


Fig.12.3.36.6 (d)

2. When tool setting of the each tool is set as Table12.3.36.6 (c), tip figure is made as Fig.12.3.36.6 (e).

Table12.3.36.6 (c)

Tool type	Tool setting
General-purpose tool	5 6 7 8 13 14 15 16
Threading tool	5 6 7 8 13 14 15 16

Tool type	Tool setting
Grooving tool	
Round-nose tool	
Point nose straight tool	
Multifunctional tool	

- Thickness of the tip (a) : The opposite direction of adjacent vertex 1 of figure element 1
- Length of the tip (b) : The same direction of adjacent vertex 2 of figure element 1
- Width of the tip (c) : The opposite direction of adjacent vertex 3 of figure element 1

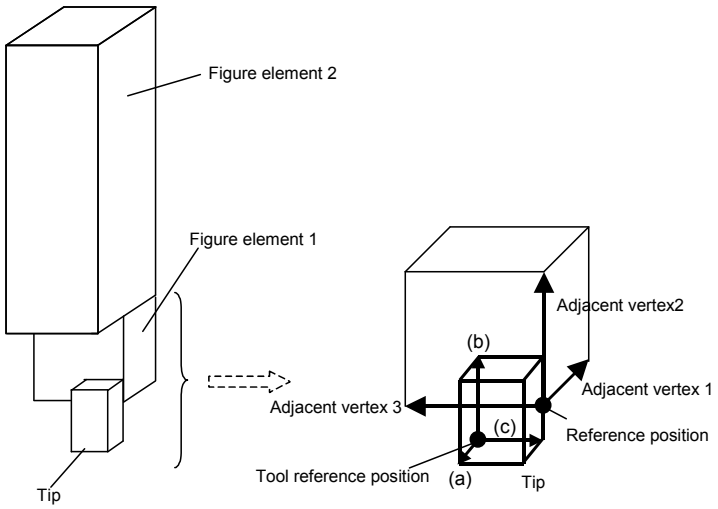
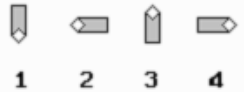


Fig.12.3.36.6 (e)

3. When tool setting of point nose straight tool is set as Table12.3.36.6 (d), tip figure is made as Fig.12.3.36.6 (f).

Table12.3.36.6 (d)

Tool type	Tool set
Point nose straight tool	

- Thickness of the tip (a) : The opposite direction of adjacent vertex 1 of figure element 1
- Length of the tip (b) : The same direction of adjacent vertex 2 of figure element 1
- Width of the tip (c) : The both same and opposite direction of adjacent vertex 3 of figure element 1

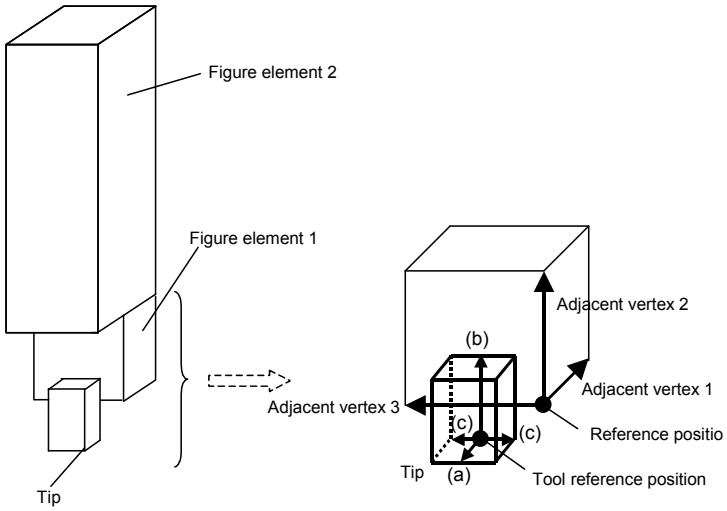


Fig.12.3.36.6 (f)

NOTE
When tool setting is set to 0, the method of making tip figure is the same as conventional method in the past.

Making method 2

The following tools are made in the cylinder as Fig.12.3.36.6 (g).

- Drill
- Flat end mill
- Ball end mill
- Tap
- Reamer
- Boring tool
- Face mill
- Angle tool
- Side tool for drill

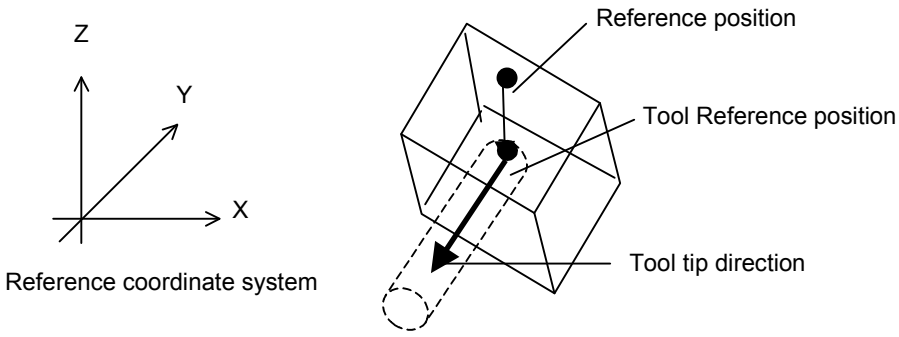
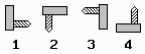
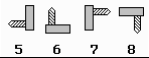


Fig.12.3.36.6 (g)

NOTE
- The tool tip direction determines the tool axis direction at machining time. The plane selection commands (G17, G18, G19) do not automatically change the axis direction.
- For the side tool for drill, when tool setting of the tool geometry size data is set to 

, tool direction is the same as tool tip direction. When tool setting is set to 

, tool direction is opposite in tool tip direction.

Making method 3 (Side tool for thread)

The figure of the side tool for thread is composed of three figures (figure element 1, 2 and tip figure) (Note 1).
The making method is different according to the shape of figure element 1 of the tool holder (Note 2).
Each figure is defined as follows.

1. In case that figure element 1 of tool holder is rectangular parallelepiped

Figure element 1 and 2 are defined as rectangular parallelepiped.

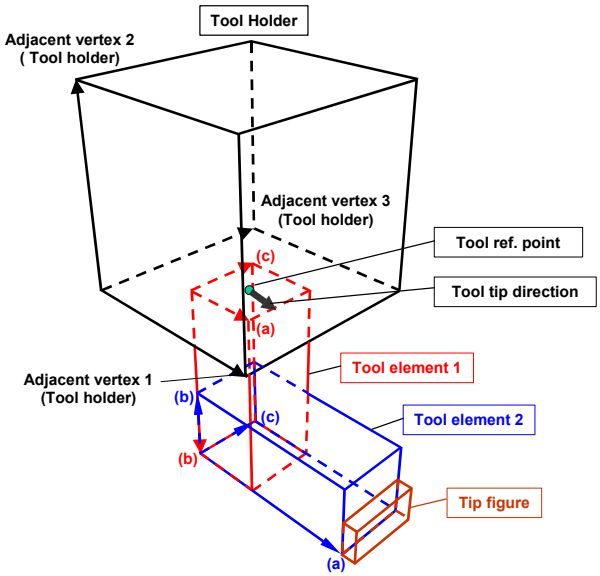


Fig.12.3.36.6 (h)

The reference vertex and the direction of vector to the adjacent vertex are set as follows.

- The tool reference position: Set center position of upper surface of tool figure element 1
- Figure element 1
  - The reference vertex: Calculated by tool reference position and tool tip direction that are set to tool holder automatically (Fig.12.3.36.6 (i))

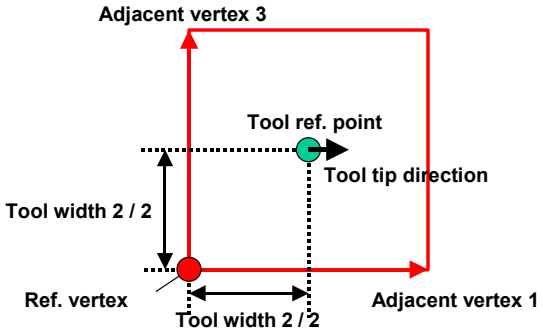


Fig.12.3.36.6 (i)

- Adjacent vertex 1 (a): Tool tip direction set to tool holder (Length = Tool width 2)
- Adjacent vertex 2 (b): The opposite direction to adjacent vertex 2 of tool holder (Length = Tool length 1)
- Adjacent vertex 3 (c): Perpendicular direction to adjacent vertex 1 and 2 (Length = Tool width 2)

- Figure element 2
  - The reference vertex: Calculated as Fig.12.3.36.6 (j).

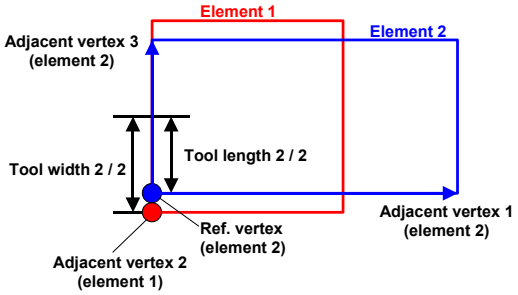


Fig.12.3.36.6 (j)

- Adjacent vertex 1 (a): The same direction to adjacent vertex 1 of figure element 1 (Length = Tool width 1)
- Adjacent vertex 2 (b): The opposite direction to adjacent vertex 2 of figure element 1 (Length = Tool length 2)
- Adjacent vertex 3 (c): The same direction to adjacent vertex 3 of figure element 1 (Length = Tool length 2)

- Tip figure

The reference position and each figure data of tip figure are set as follows.

- The reference position: Center of side of element 2
- Length of the tip: The opposite direction from reference vertex to adjacent vertex 2 of figure element 2 (Tip width 1)
- Width of the tip: The same direction from reference vertex to adjacent vertex 1 of figure element 1 (Tip width 2)
- Thickness of the tip: Perpendicular direction to length and width (Tip thickness)

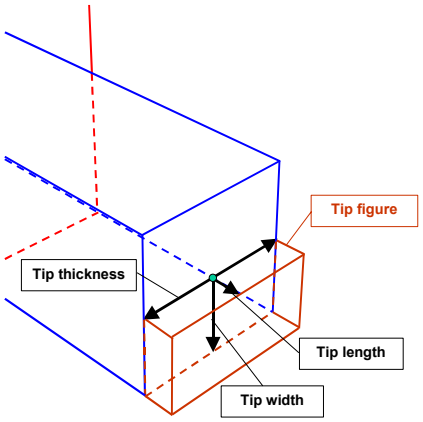


Fig.12.3.36.6 (k)

2. In case that figure element 1 of tool holder is cylinder
Figure element 1 and 2 are defined as the cylinder.

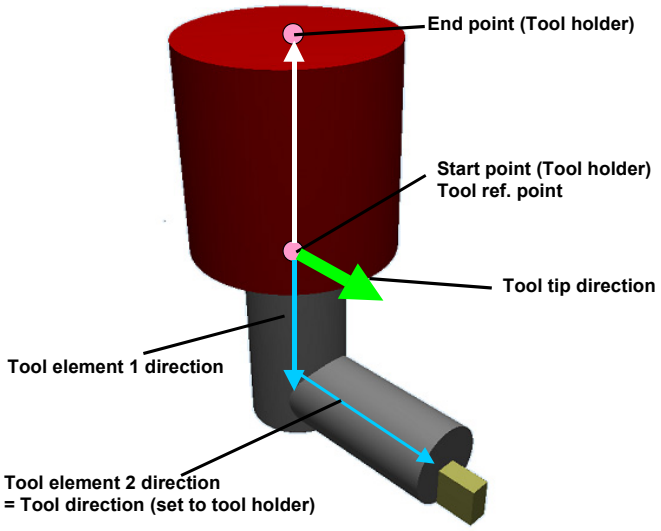


Fig.12.3.36.6 (l)

The start point, the end point and the radius of the each figure element are set as follows.

- Figure element 1
  - The start point: Tool reference point set to tool holder
  - The end point: The opposite direction from start point to end point of element 1 of tool holder (Length = Tool length 1)
  - The radius: Half of tool width 2
- Figure element 2
  - The start point: Calculated as Fig.12.3.36.6 (m)

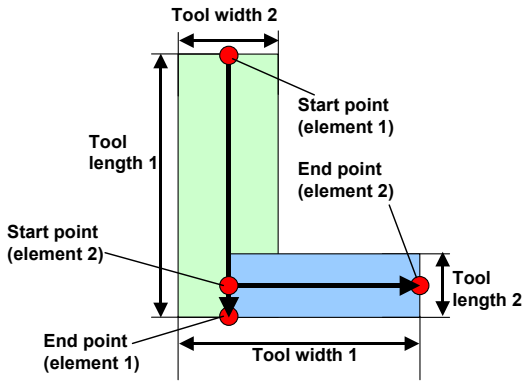


Fig.12.3.36.6 (m)

- The end point: Tool tip direction set to tool holder
  (Length = Tool width 1 - Radius of element 1)
- The radius: Half of tool length 2

- Tip figure

  The reference position and each figure data of tip figure are set as follows.
  - The reference position: The end point of figure element 2
  - Length of the tip: Tool tip direction set to tool holder (Tip width 1)
  - Width of the tip: The same direction from start point to end point of figure element 1 (Tip width 2)
  - Thickness of the tip: Perpendicular direction to length and width (Tip thickness)

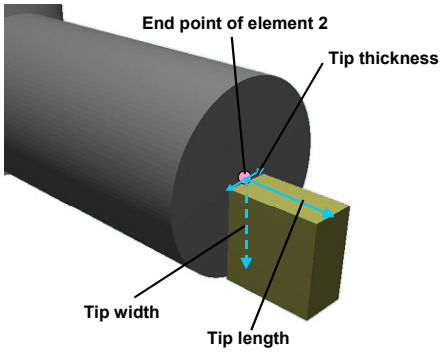
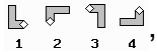


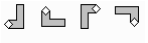
Fig.12.3.36.6 (n)

NOTE

1 When you use the side tool for thread, the number of figures that can be defined by other figure decreases. For example, when the side tool is used in 1-path, the number of figure that can be defined by other figure is 21 (=23-2). When the side tool is used at the same time in multi-path system, the maximum number of figures that can be defined decreases further.

2 When figure element 1 is defined by the plane or it is not defined, tool figure is not made. (Alarm PS0492 “3DCHK FIG. ILLEGAL: TOOL” is issued.)

3 When tool setting of the tool geometry size data is set to 

, tool figure is made in the direction of tool tip direction. When tool setting is set to 

, tool figure is made in the opposite direction of tool tip direction.

Making method 4 (Back tool)

The figure of the back tool is composed of four figures (figure element 1 to 3 and tip figure) (Note 1).
The making method is different by the shape of figure element 1 of the tool holder (Note 2).

Each figure is defined as follows.

1. In case that figure element 1 of tool holder is rectangular parallelepiped
Figure element 1, 2 and 3 are defined as rectangular parallelepiped.

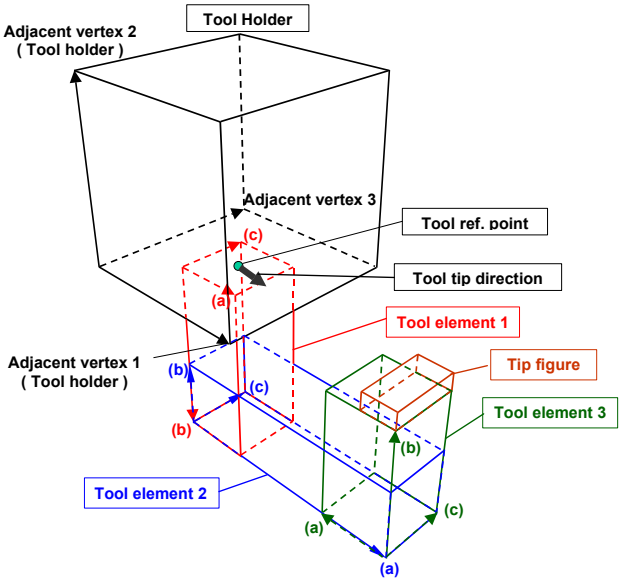


Fig.12.3.36.6 (o)

The reference position and the direction of vector to the adjacent vertex are set as follows.

- The tool reference position: Set center position of upper surface of tool figure element 1
- Figure element 1
  - The reference vertex: Calculated by tool reference position and tool tip direction that are set to tool holder automatically. (Fig.12.3.36.6 (p))

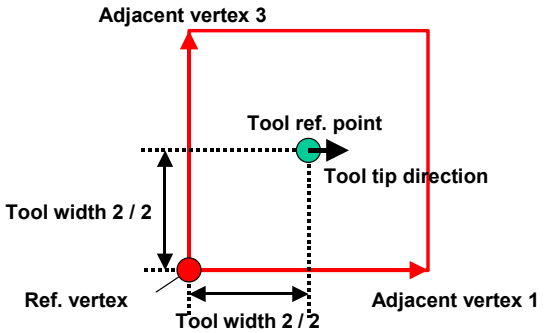


Fig.12.3.36.6 (p)

  - Adjacent vertex 1 (a): Tool tip direction set to tool holder. (Length = Tool width 2)
  - Adjacent vertex 2 (b): The opposite direction to adjacent vertex 2 of tool holder. (Length = Tool length 1)
  - Adjacent vertex 3 (c): Perpendicular direction to adjacent vertex 1 and 2 (Length = Tool width 2)
- Figure element 2
  - The reference vertex: Adjacent vertex 2 of figure element 1
  - Adjacent vertex 1 (a): The same direction to adjacent direction 1 of figure element1 (Length = Tool width 1)
  - Adjacent vertex 2 (b): The opposite direction to adjacent vertex 2 of figure element 1 (Length = Tool width 2)

- Adjacent vertex 3 (c): The same direction to adjacent vertex 3 of figure element 1
  (Length = Tool width 2)

- Figure element 3
  - The reference vertex: Calculated as Fig.12.3.36.6 (q)

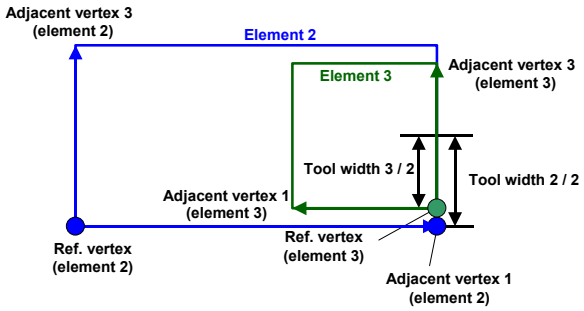


Fig.12.3.36.6 (q)

- Adjacent vertex 1 (a): The opposite direction to adjacent vertex 1 of figure element 1
  (Length = Tool width 3)
- Adjacent vertex 2 (b): The opposite direction to adjacent vertex 2 of figure element 1
  (Length = Tool length 2)
- Adjacent vertex 3 (c): The same direction to adjacent vertex 3 of figure element 1
  (Length = Tool width 3)

- Tip figure
  The reference position and each figure data of tip figure are set as follows.
  - The reference position: Center position of upper surface of tool figure element 3
  - Length of the tip: The opposite direction to adjacent vertex 2 of figure element 2
    (Tip width 1)
  - Width of the tip: The same direction to adjacent vertex 1 of figure element 1
    (Tool width 2)
  - Thickness of the tip: Perpendicular direction to length and width (Tip thickness)

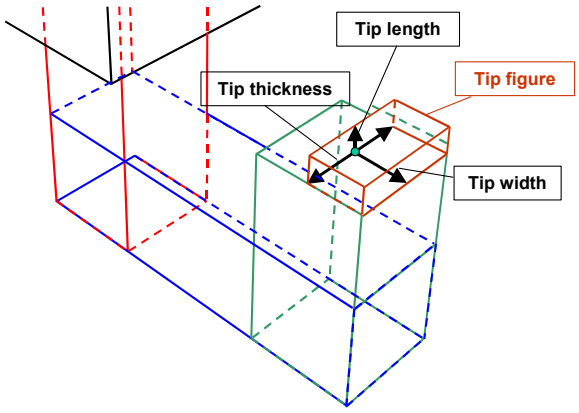


Fig.12.3.36.6 (r)

2. In case that figure element 1 of tool holder is cylinder
Figure element 1, 2 and 3 are defined as the cylinder.

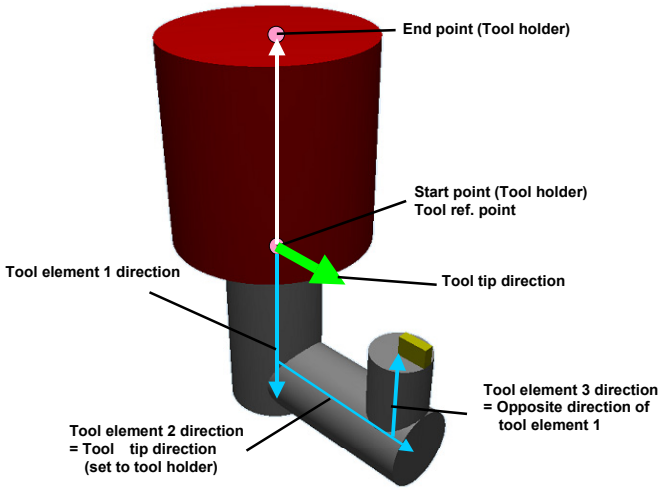


Fig.12.3.36.6 (s)

The start point, the end point and the radius of the each figure element are set as follows.

- Figure element 1
  - The start point: Tool reference position set to tool holder
  - The end point: The opposite direction from start point to end point of element 1 of tool holder (Length = Tool length 1)
  - The radius: Half of tool width 2
- Figure element 2
  - The start point: Calculated as Fig.12.3.36.6 (t)
  - The end point: Tool tip direction set to tool holder (Length = Tool width 1 - Radius of element 1)
  - The radius: The same as the radius of figure element 1
- Figure element 3
  - The start point: Calculated as Fig.12.3.36.6 (t)
  - The end point: The opposite direction from start point to end point of element 1 (Length = Tool length 2 - Radius of element 2)
  - The radius: Half of tool width 3

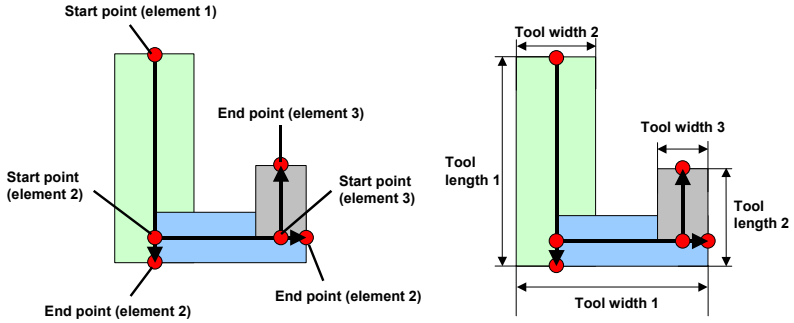


Fig.12.3.36.6 (t)

- Tip figure
  The reference position and each figure data of tip figure are set as follows.
  - The reference position: The end point of figure element 3
  - Length of the tip: The opposite direction from start point to end point of figure element 1 (Tip width 1)
  - Width of the tip: The same direction from start point to end point of figure element 2 (Tip width 2)
  - Thickness of the tip: Perpendicular direction to length and width (Tip thickness)

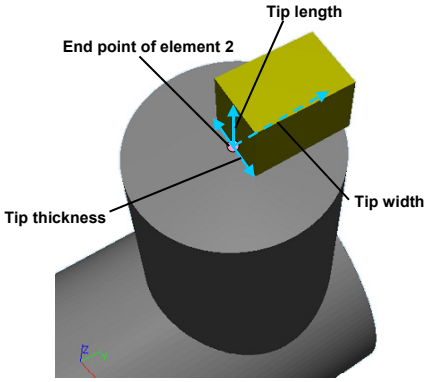
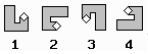


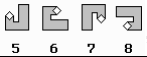
Fig.12.3.36.6 (u)

NOTE

1 When you use the back tool, the number of figures that can be defined by other figure decreases. For example, when the back tool is used in 1-path, the number of figure that can be defined by other figure is 20 (=23-3). When the back tool is used at the same time in multi-path system, the maximum number of figures that can be defined decreases further.

2 When figure element 1 is defined by the plane or it is not defined, tool figure is not made. (Alarm PS0492 “3DCHK FIG. ILLEGAL: TOOL” is issued.)

3 When tool setting of the tool geometry size data is set to 

, tool figure is made in the direction of tool tip direction. When tool setting is set to 

, tool figure is made in the opposite direction of tool tip direction.

Shape type display

To the right of each shape number, the icon indicating the shape type selected as the figure element is displayed.
Each icon indicates a shape type as follows:


: Rectangular parallelepiped
: Cylinder
: Plane

When shape data is not set, no shape type icon is displayed.

Operation

On the tool holder figure setting screen, the following soft keys are available to perform operation:

[EDIT]	Edits shape data for the shape number selected by the cursor.
[SHAPE LIST]	Displays the shape number list screen.
[MENU]	Displays the figure setting menu screen.
[CHECK]	Updates the display for checking the input shape.
[INPUT]	Inputs data. Performs the same operation as MDI key  .
[ROTATIION]	Rotates the figure for check.
[SHAPE SET]	Sets a new shape or changes the shape type.
[FIGURE NO.SRH]	Displays the figure setting screen for the input figure number.

Pressing soft key [ROTATION] causes soft keys for rotation operation to appear.



Fig. 12.3.36.6 (v) Soft keys for rotation operation

Soft key [↑] Rotates the figure for check upward.
Soft key [↓] Rotates the figure for check downward.
Soft key [←] Rotates the figure for check to left.
Soft key [→] Rotates the figure for check to right.

Procedure for setting new shape data

To set new shape data, use the following procedure:

1. Move the cursor to the SHAPE NO. field for a figure element you want to set.
2. Enter a shape number and press the INPUT key or soft key [INPUT].
3. Press soft key [SHAPE SET]. Pressing soft key [SHAPE SET] causes the soft keys shown in Fig. 12.3.36.6 (w) to appear:



Fig. 12.3.36.6 (w) Shape type soft keys

4. Press the soft key corresponding to the shape type you want to set. Pressing the soft key selects the relevant shape type and displays the relevant setting screen.

Procedure for changing the shape type

To change the shape type of set shape data, use the following procedure:

1. Move the cursor to the SHAPE NO. field for the figure element of which shape type you want to change.

2. Press soft key [SHAPE SET]. Pressing soft key [SHAPE SET] causes the following soft keys to appear:
   At this time, the soft key corresponding to the currently set shape type does not appear.



Fig. 12.3.36.6 (x) Soft keys displayed when the current setting is rectangular parallelepiped



Fig. 12.3.36.6 (y) Soft keys displayed when the current setting is cylinder



Fig. 12.3.36.6 (z) Soft keys displayed when the current setting is plane

3. Press the soft key corresponding to the shape type you want to set. Pressing the soft key causes a confirmation message and the soft keys shown in Fig. 12.3.36.6 (aa) to appear:

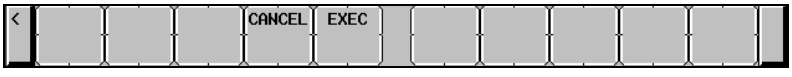


Fig. 12.3.36.6 (aa) Soft keys displayed after a shape type is selected

4. Pressing soft key [EXEC] causes the setting screen for the selected shape type to appear.
   The shape data before change is cleared.
   To cancel the change of the shape type, press soft key [CANCEL].

Procedure for editing shape data

To edit shape data, use the following procedure:

1. Move the cursor to the SHAPE NO. field for a figure element you want to edit.

2. Press soft key [EDIT]. Pressing soft key [EDIT] causes the setting screen for the set shape type to appear. When no shape type is set, the rectangular parallelepiped setting screen appears.

12.3.36.7 Rectangular parallelepiped setting screen

Screen configuration

Define a rectangular parallelepiped as a figure element. To define a rectangular parallelepiped, set the following five data items:

- Reference vertex : Position of the reference vertex viewed from the reference position
- Adjacent vertex 1: Position of adjacent vertex 1 viewed from to the reference vertex
- Adjacent vertex 2: Position of adjacent vertex 2 viewed from to the reference vertex
- Adjacent vertex 3: Position of adjacent vertex 3 viewed from to the reference vertex
- Numerical unit : Machine unit or input unit

The shape corresponding to the input data is displayed on the right side of the screen.
To check the shape, set or change the above five data items and press soft key [CHECK]. The display for checking the shape is updated.
When rectangular parallelepiped data is not input or is invalid, no shape is displayed.

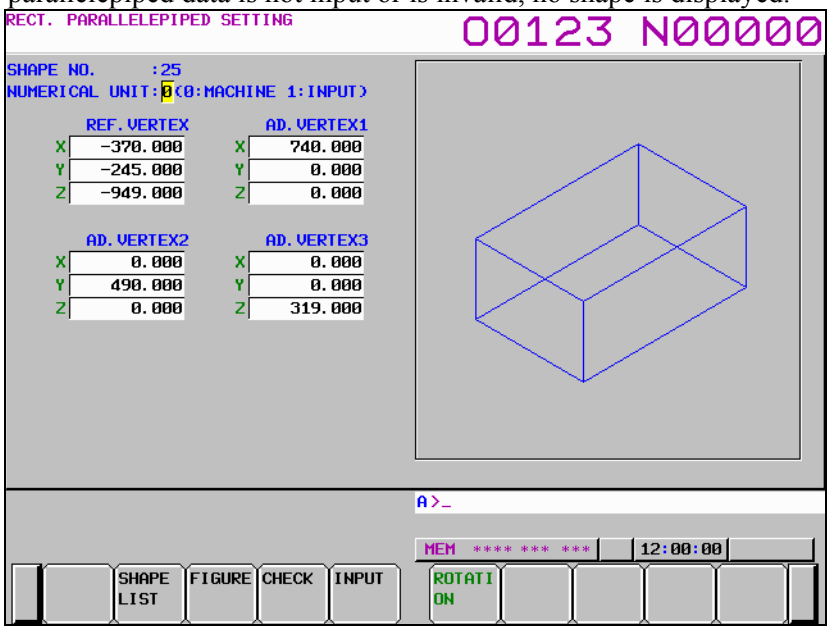


Fig. 12.3.36.7 (a) Rectangular parallelepiped setting screen 10.4”

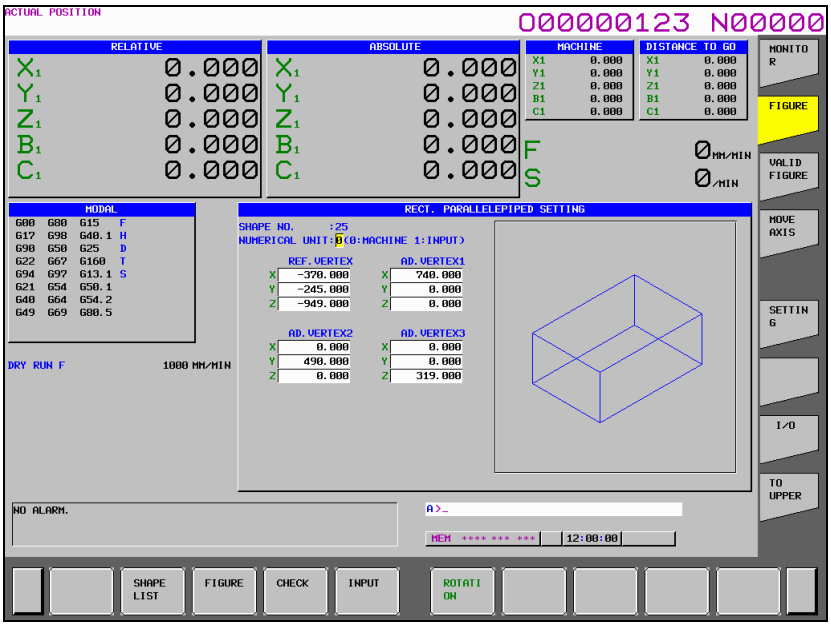


Fig. 12.3.36.7 (b) Rectangular parallelepiped setting screen 15”

Operation

On the rectangular parallelepiped setting screen, the following soft keys are available to perform operation:

[SHAPE LIST] Displays the shape number list screen.
[FIGURE] Returns to the figure setting screen.
[CHECK] Updates the display for checking the input shape.
[INPUT] Inputs data. Performs the same operation as MDI key <INPUT>.
[ROTATION] Rotates the shape for check.

Pressing soft key [ROTATION] causes soft keys for rotation operation to appear.



Fig. 12.3.36.7 (c) Soft keys for rotation operation

Soft key [↑] Rotates the shape for check upward.
Soft key [↓] Rotates the shape for check downward.
Soft key [←] Rotates the shape for check to left.
Soft key [→] Rotates the shape for check to right.

> ⚠ **CAUTION**
> The setting you make is not updated until you turn the power off, then on again, or set built-in 3D interference check setting change signal TDICHG to “1”.

12.3.36.8 Cylinder setting screen

Screen configuration

Define a cylinder as a figure element. To define a cylinder, set the following four data items:

- Start point : Position of the start point of the cylinder, viewed from the reference position
- End point : Position of the end point, viewed from the start point of the cylinder
- Radius : Radius of the cylinder
- Numerical unit: Machine unit or input unit

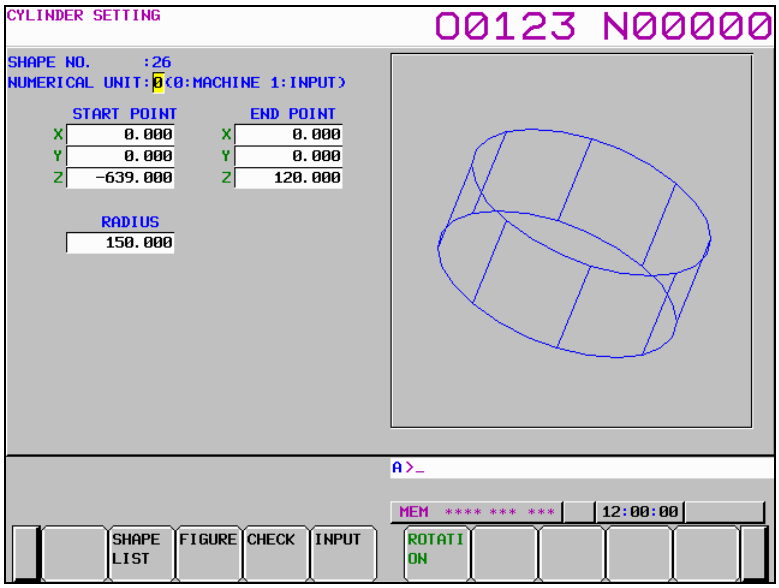


Fig. 12.3.36.8 (a) Cylinder setting screen 10.4”

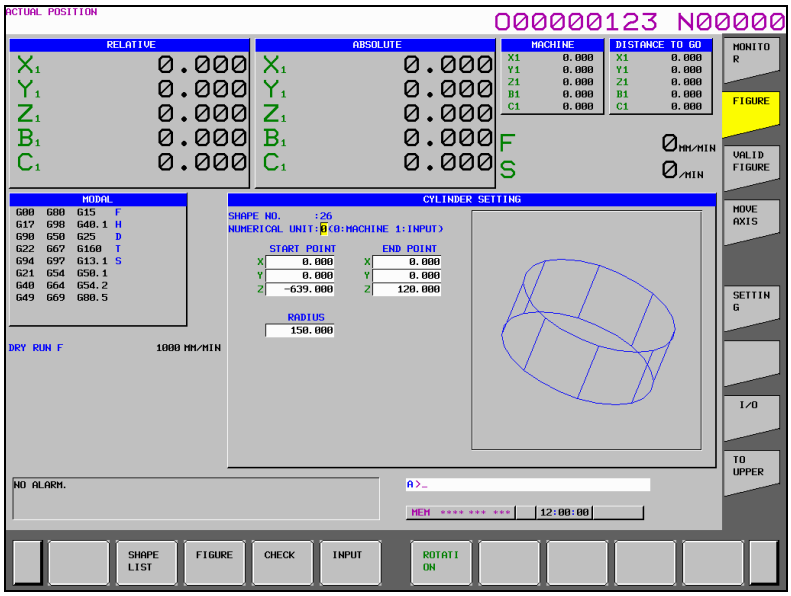


Fig. 12.3.36.8 (b) Cylinder setting screen 15”

Operation

On the cylinder setting screen, the following soft keys are available to perform operation:

[SHAPE LIST] Displays the shape number list screen.
[FIGURE] Returns to the figure setting screen.
[CHECK] Updates the display for checking the input shape.
[INPUT] Inputs data. Performs the same operation as MDI key .
[ROTATION] Rotates the shape for check.

12.3.36.9 Plane setting screen

Screen configuration

Define a plane as a figure element. To define a plane, set the following three data items:

- Point on plane : Position of an arbitrary point on the plane, viewed from the reference position

- Normal vector : Vector perpendicular to the plane
- Numerical unit : Machine unit or input unit

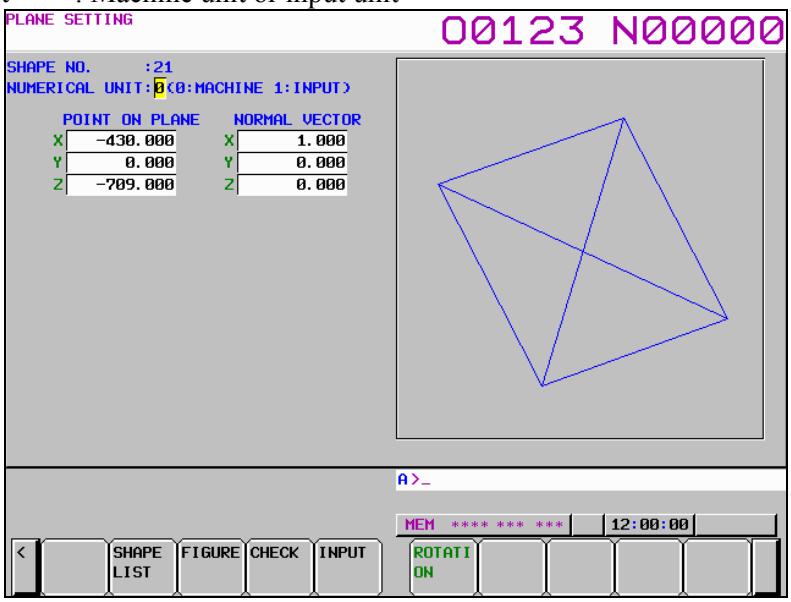


Fig. 12.3.36.9 (a) Plane setting screen 10.4”

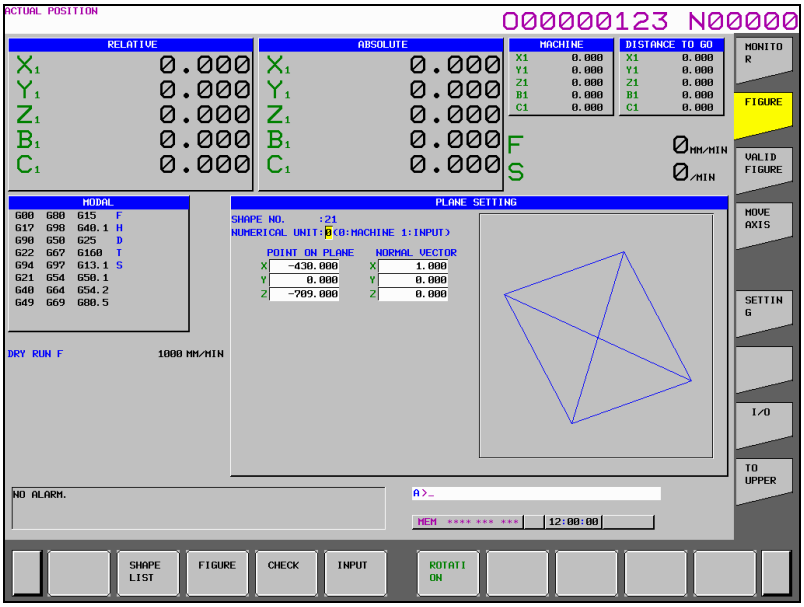


Fig. 12.3.36.9 (b) Plane setting screen 15”

Operation

On the plane setting screen, the following soft keys are available to perform operation:

[SHAPE LIST] Displays the shape number list screen.
[FIGURE] Returns to the figure setting screen.
[CHECK] Updates the display for checking the input shape.
[INPUT] Inputs data. Performs the same operation as MDI key 

.
[ROTATION] Rotates the shape for check.

NOTE
As the shape for check, a plane of a fixed size of which center is the arbitrary point is displayed. For interference check, an infinite-size plane is used.

12.3.36.10 Shape number list screen

The shape number list screen displays a list of the figure names and figure numbers assigned to shape numbers. This list screen is the same as the shape number list screen displayed from the tool holder figure setting screen described before. When a shape is assigned to multiple figures, the names and numbers of the figures are all displayed. When a shape is assigned to no figure, the FIGURE NAME and FIG. NO. fields are left blank.
If shape data is not set or is invalid, an asterisk (*) is displayed to the left of the shape number.
Up to 150 shape numbers can be used. The number of displayed shapes can be decreased by setting the number of shapes on the relevant setting screen, which can hide the figure data items having numbers greater than the set number.

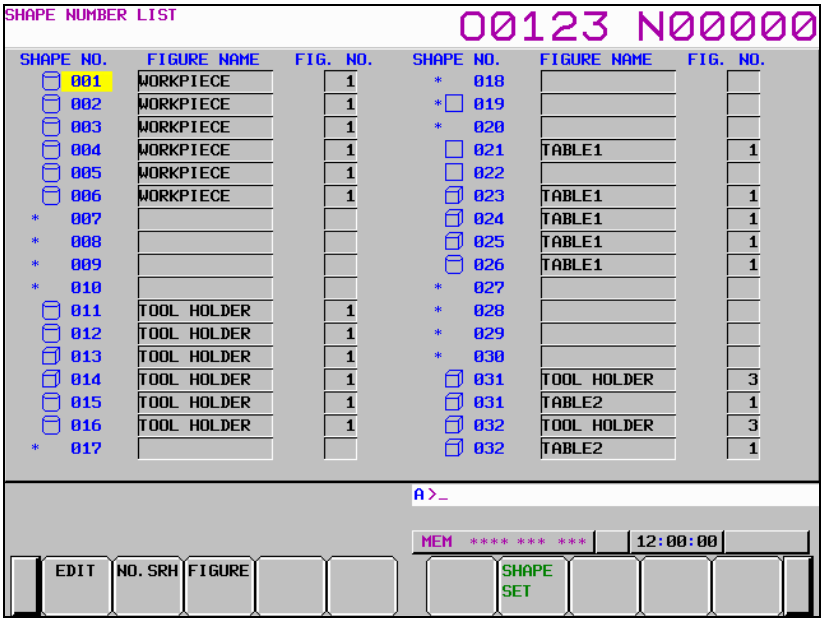


Fig. 12.3.36.10 (a) Shape number list screen 10.4”

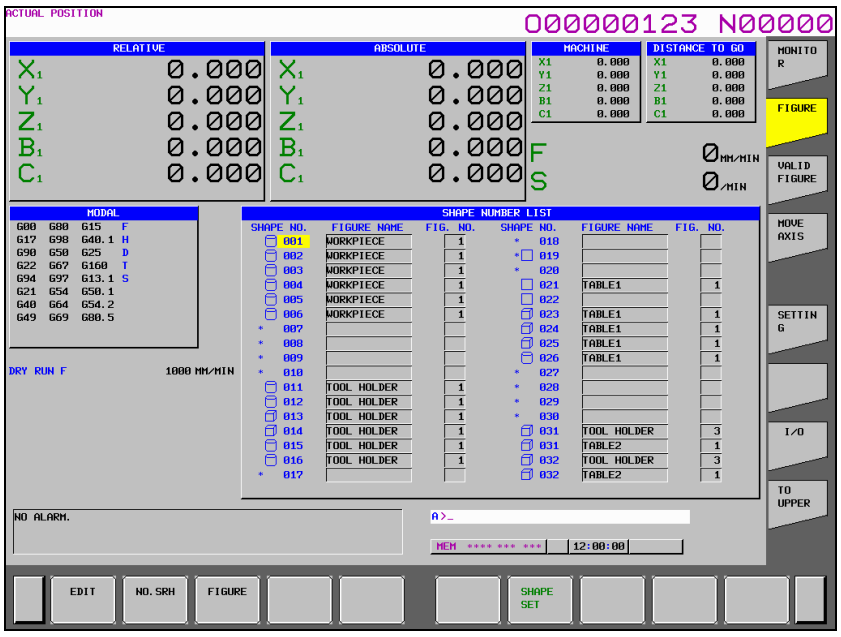


Fig. 12.3.36.10 (b) Shape number list screen 15”

Moving the cursor on a shape number and pressing soft key [EDIT] causes the setting screen for the relevant shape to appear to enable the editing of the setting. In Fig. 12.3.36.10 (a) and Fig. 12.3.36.10 (b), data of shape number 1 can be edited.

Shape type display

To the left of each shape number, the icon indicating the shape type is displayed.


: Rectangular parallelepiped
: Cylinder
: Plane

When shape data is not set, no shape type icon is displayed.

Operation

On the shape number list screen, the following soft keys are available to perform operation:

[EDIT]	Edits shape data for the shape number selected by the cursor.
[NO.SRH]	Searches for a shape number input in the key-in buffer.
[FIGURE]	Displays the figure setting screen.
[SHAPE SET]	Sets a new shape or changes the shape type.

Procedure for setting new shape data

To set new shape data, use the following procedure:

1. Move the cursor to the SHAPE NO. field for a figure element you want to set.
2. Press soft key [SHAPE SET]. Pressing soft key [SHAPE SET] causes the soft keys shown in Fig. 12.3.36.10 (c) to appear:



Fig. 12.3.36.10 (c) Shape type soft keys

3. Press the soft key corresponding to the shape type you want to set. Pressing the soft key selects the relevant shape type and displays the relevant setting screen.

Procedure for changing the shape type

To change shape data, use the following procedure:

1. Move the cursor to the shape number of which data you want to change.

2. Press soft key [SHAPE SET]. Pressing soft key [SHAPE SET] causes the following soft keys to appear:
   At this time, the soft key corresponding to the currently set shape type does not appear.



Fig. 12.3.36.10 (d) Soft keys displayed when the current setting is rectangular parallelepiped



Fig. 12.3.36.10 (e) Soft keys displayed when the current setting is cylinder



Fig. 12.3.36.10 (f) Soft keys displayed when the current setting is plane

3. Press the soft key corresponding to the shape type you want to set. Pressing the soft key causes a confirmation message and the soft keys shown in Fig. 12.3.36.10 (g) to appear:

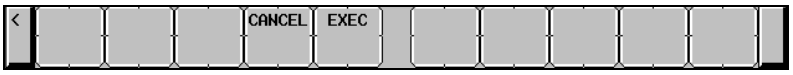


Fig. 12.3.36.10 (g) Soft keys displayed after a shape is selected

4. Pressing soft key [EXEC] causes the setting screen for the selected shape type to appear.
   The shape data before change is cleared.
   To cancel the change of the shape type, press soft key [CANCEL].

Procedure for editing shape data

To edit shape data, use the following procedure:

1. Move the cursor to the shape number of which data you want to edit.

2. Press soft key [EDIT]. Pressing soft key [EDIT] causes the setting screen for the set shape type to appear. When no shape type is set, the rectangular parallelepiped setting screen appears.

12.3.36.11 Interference check valid figure selection screen

On the interference check valid figure selection menu screen, select a target item in the same way as on the figure setting menu screen.
To display the interference check valid figure selection screen, use the following procedure:

1. Press function key .
2. Press soft key [3D INTER.].
3. Press soft key [VALID FIGURE].

To display the interference check valid figure selection screen, select an object on the menu screen and press soft key [SELECT].
To define a valid figure, select a figure for which to enable 3D interference check as shown below:

- Interference check valid figure:
  Specify a figure number (0 to 10) for which to enable 3D interference check. When a value of 0 is specified, the following setting is made according to the setting of bit 3 (ICV) of parameter No. 10930 as follows:
  0: Sets figure 1 as the valid figure.
  1: Assumes that there is no figure. (Does not perform interference check.)

You can check the currently selected figure on the monitor screen.
This screen is used only for objects.

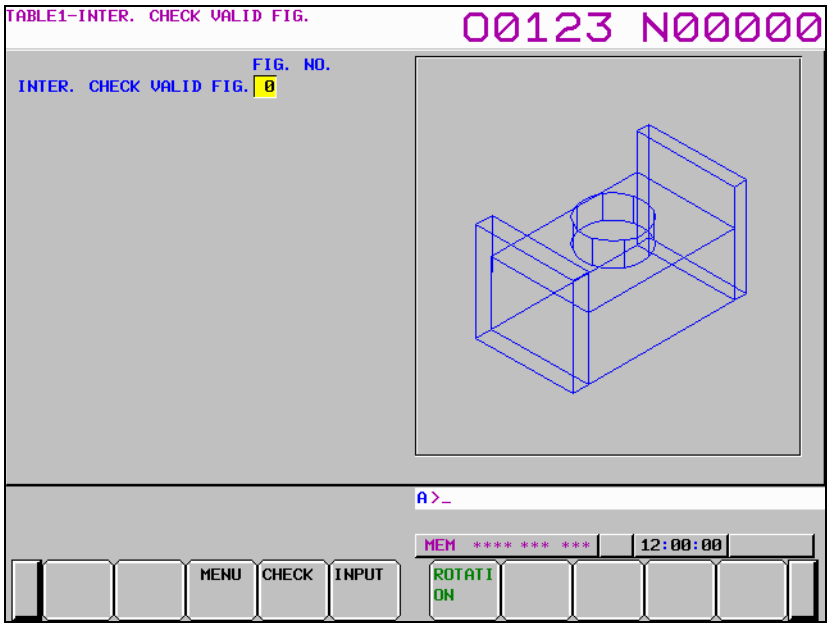


Fig. 12.3.36.11 (a) Interference check valid figure selection screen 10.4”

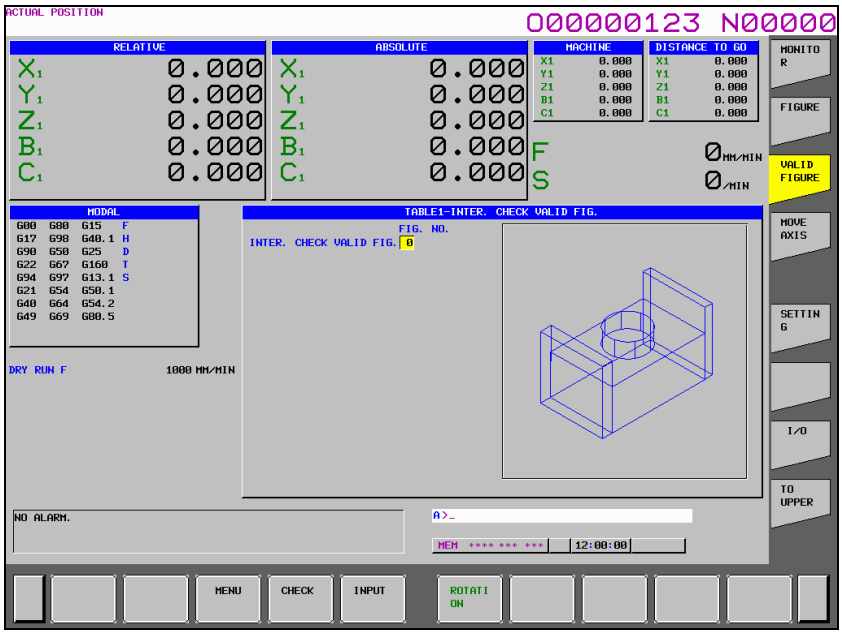


Fig. 12.3.36.11 (b) Interference check valid figure selection screen 15”

In the setting example shown in Fig. 12.3.36.11 (a) and Fig. 12.3.36.11 (b), 3D interference check is made for the object having figure number 1. Interference check is not made for other than figure 1. You can change the valid figure number using G22.2, however.

If no figure element (rectangular parallelepiped, cylinder, and plane) is defined for the selected figure, alarm PS0492 “3DCHK FIG. ILLEGAL: [Target name]” is issued.

> ⚠ **CAUTION**
> The setting you make is not updated until you turn the power off, then on again, or set built-in 3D interference check setting change signal TDICHG to “1”.

12.3.36.12 Moving axis setting menu screen and moving axis setting screen

On the moving axis setting menu screen, select a target item in the same way as on the figure setting menu screen.
To display the moving axis setting menu screen, use the following procedure:

1. Press function key .
2. Press soft key [3D INTER.].
3. Press soft key [MOVE AXIS].

To display the moving axis setting screen, select a tool holder or object on the menu screen and press soft key [SELECT].
Set items including linear axes along which to and rotation axes about which to move each object or tool holder.
To define a moving axis, set the following data items:

1) Synchronous object number:
   Set the number of another object (1 to 6) moving in synchronization with the target object if any. Set 0 when no object moves in synchronization with the target object.
   This item is not displayed for a tool holder.
2) Absolute path number:
   Set the path to which the object belongs. Set 0 for path 1.
   This item is not displayed for a tool holder.
3) Axis number of the first, second, and third linear axes:
   Set the axis number of the axis parallel to the X-, Y-, or Z-axis in the reference coordinate system as the axis number of each of the 1st, 2nd, and 3rd linear axes along which to move the target item. When the relevant moving axis is not used, set 0. In the direction field, set 0 when the positive direction of the 1st, 2nd, or 3rd linear axis is the same as that of the corresponding X-, Y-, or Z-axis in the reference coordinate system or 1 when the direction is opposite to the positive direction of the corresponding axis.
4) Axis number of the first and second rotation axes:
   Set the axis number of each of the 1st and 2nd rotation axes about which to rotate the target item. When the relevant rotation axis is not used, set 0.
5) Rotation center axis directions of the first and second rotation axes:(Note 1)
   Set the rotation center axis direction, viewed from the reference coordinate system, of each of the 1st and 2nd rotation axes about which to rotate the target item.
6) Inclination angles of the first and second rotation axes:(Note 2)
   Set the inclination angle when the 1st or 2nd rotation axis about which to rotate the target item is an inclined rotation axis.
7) Rotation center axis positions of the first and second rotation axes:
   Set the position of the rotation center axis, viewed from the reference coordinate system, of each of the 1st and 2nd rotation axes about which to rotate the target item.
8) Rotation directions of the first and second rotation axes:
   Set the rotation direction of each of the 1st and 2nd rotation axes about which to rotate the target item. Set 0 when the axis rotates clockwise (CW), viewed from the negative direction to the positive direction of the axis specified in 7), or 1 when the axis rotates counterclockwise (CCW).
9) Rotation of figure elements 1 to 6:
   Set 0 when figure element 1 to 6 rotates about the master or slave rotation axis or 1 when it does not rotate about the axis.

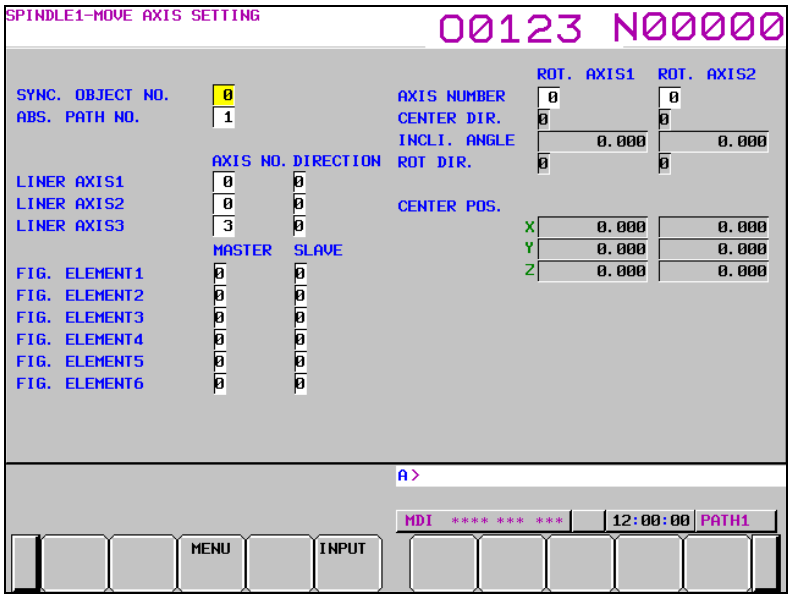


Fig. 12.3.36.12 (a) Moving axis setting screen 10.4”

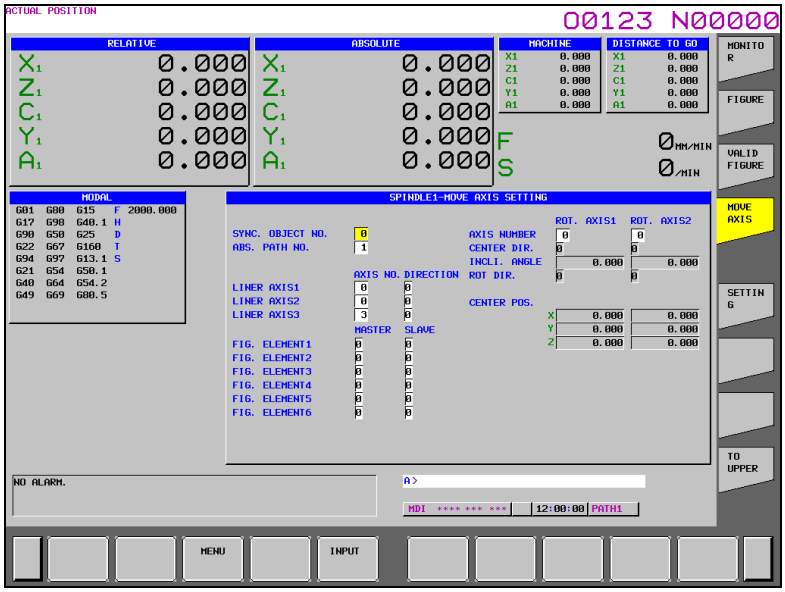
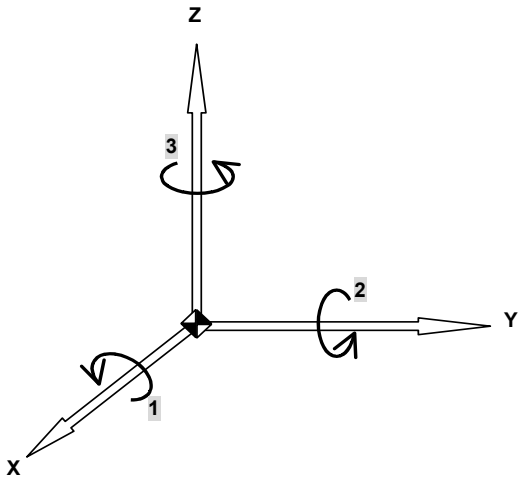


Fig. 12.3.36.12 (b) Moving axis setting screen 15”

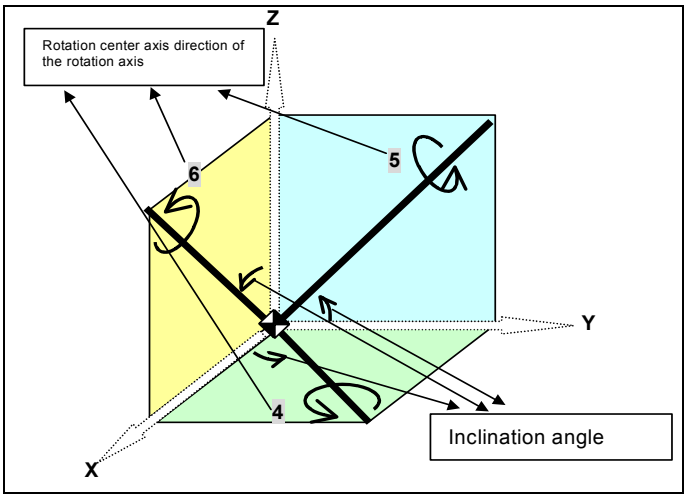
NOTE

1 Set the direction of the rotation center axis of the rotation axis.
- 1: On X-axis
- 2: On Y-axis
- 3: On Z-axis
- 4: On an axis tilted a certain angle from the X-axis from the positive X-axis to positive Y-axis
- 5: On an axis tilted a certain angle from the Y-axis from the positive Y-axis to positive Z-axis
- 6: On an axis tilted a certain angle from the Z-axis from the positive Z-axis to positive X-axis

(A value 4 to 6 is to be set when the inclined rotation axis control function is used.)



2 Set the inclination angle when the 1st or 2nd rotation axis is an inclined rotation axis.



⚠ CAUTION

The setting you make is not updated until you turn the power off, then on again, or set built-in 3D interference check setting change signal TDICHG to “1”.

12.3.36.13 Setting screens

On setting screens, set the following items:

- Names of tools, tool holders, and objects
- Number of displayed shapes
- Whether to display or hide the figure setting screen and moving axis setting screen of each of tool holders and objects
- Drawing coordinate system

To display the setting screen, use the following procedure:

1. Press function key .
2. Press soft key [3D INTER.].
3. Press soft key [SETTING].

There are three setting screens: “name setting screen”, “display setting screen”, and “drawing coordinate system setting screen”.
To display these setting screens, press soft key [SELECT].

In a multi-path system, not data for each path, but data common to the system is set.
Data may be set for any path.
When bit 1 (ICD) of parameter No. 10930 is set to 1, the setting screens are not displayed.

12.3.36.14 Name setting screen

Define the names of tools 1 to 4, tool holders 1 to 4, and objects 1 to 6. To define a name, set the following two data items:

- Number:
  For the correspondence between names and numbers, see Table 12.3.36.14 (a), “Correspondence between names and numbers”.
- Suffix:
  A suffix is a number added following a name. Use a suffix to distinguish between different items for which the number indicating the same name is specified. When the specified number is 0, no suffix is added.

When the number of controlled paths is 1, the items for object 4, object 5, and object 6 can be displayed by setting parameter ENO (No.10930#6) to 1.
The items for Tool 2, Tool holder 2 are displayed when the number of controlled paths is 2 or more.
The items for Tool 3 and Tool holder 3 are displayed when the number of controlled paths is 3 or more.
The items for Tool 4 and Tool holder 4 are displayed when the number of controlled paths is 4 or more.

Table 12.3.36.14 (a) Correspondence between names and numbers

Number	Interference check target item names		
	Tool	Tool holder	Object
0	Default name	Default name	Default name
1	“CUTTER”	“TURRET”	“SPINDLE”
2	“SINGLE PNT”	“TOOL HOLDER”	“TABLE”
3	“TOOL”	“TOOL POST”	“WORKPIECE”
4		“SPINDL HEAD”	“CHUCK”
5			“JIG”
6			“TAILSTOCK”
7			“TOOL HOLDER”
8			“STRUCTURE”
9			“OBJECT”
10			“WORK HEAD”

Number	Interference check target item names		
	Tool	Tool holder	Object
11			"WORK LIMIT"

Default names

The default names listed in Table 12.3.36.14 (b) below are used.

In a combined system in multi-path control, the default names of the machine control type for each path are used for tools and tool holders. For objects, however, when the M series is set in all paths, the default names for the M series are used; when the T series is set in at least one path, the default names for the T series are used.

Table 12.3.36.14 (b) Default name

NC system type	Default name			
	Tool	Tool holder	Object	
T series	"CUTTER"	"TURRET"	Object 1	"WORKPIECE"
			Object 2	"CHUCK"
			Object 3	"SPINDLE"
			Object 4	"WORKPIECE"
			Object 5	"CHUCK"
			Object 6	"SPINDLE"
M series	"TOOL"	"TOOL HOLDER"	Object 1	"WORKPIECE"
			Object 2	"JIG"
			Object 3	"TABLE"
			Object 4	"WORKPIECE"
			Object 5	"JIG"
			Object 6	"TABLE"

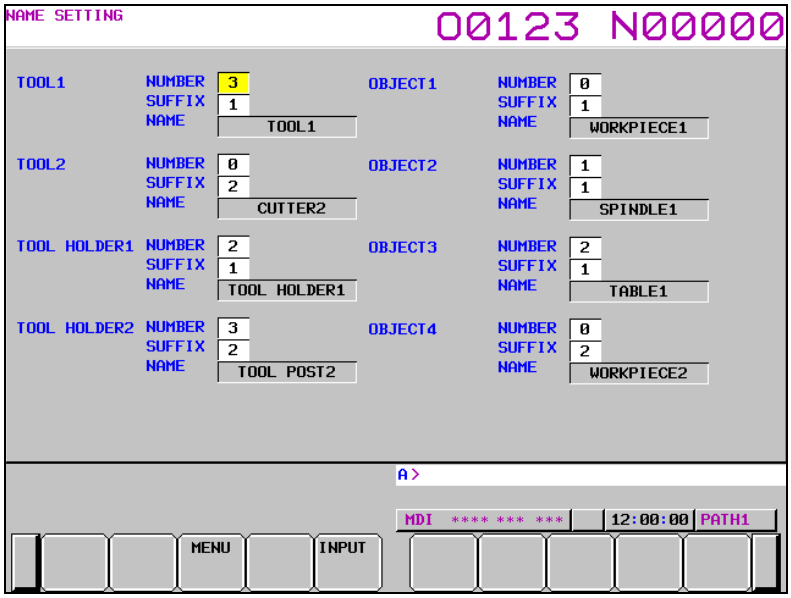


Fig. 12.3.36.14 (a) Name setting screen 10.4"

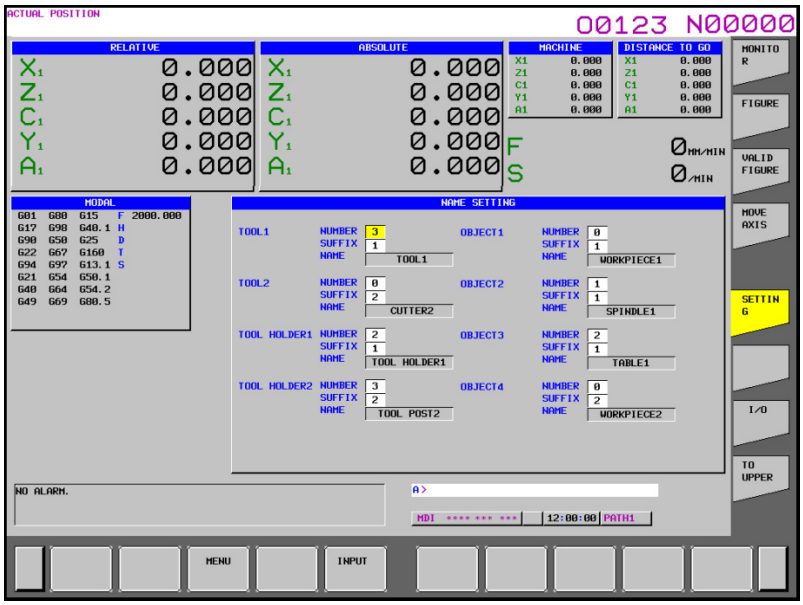


Fig. 12.3.36.14 (b) Name setting screen 15”

12.3.36.15 Display setting screen

Define the items displayed on built-in 3D interference check function setting screens. Set the following items:

- Number of shapes:
  Set a numeric value between 0 and 150. On the shape number list screen, the numbers between 0 and the set number are displayed. The numbers greater than the set number are not displayed. For any non-displayed number, data cannot be set on the figure setting screen. When a value of 0 is set, 150 is assumed.
- Displaying/hiding the move axis setting/figure setting screens:
  Select whether to display or hide the figure setting screen and moving axis setting screen of each of tool holders and objects.
  When the number of controlled paths is 1, the items for object 4, object 5, and object 6 can be displayed by setting parameter ENO (No.10930#6) to 1.
  The items for tool 2, and tool holder 2 are displayed when the number of controlled paths is 2 or more.
  The items for Tool holder 3 are displayed when the number of controlled paths is 3 or more.
  The items for Tool holder 4 are displayed when the number of controlled paths is 4 or more.

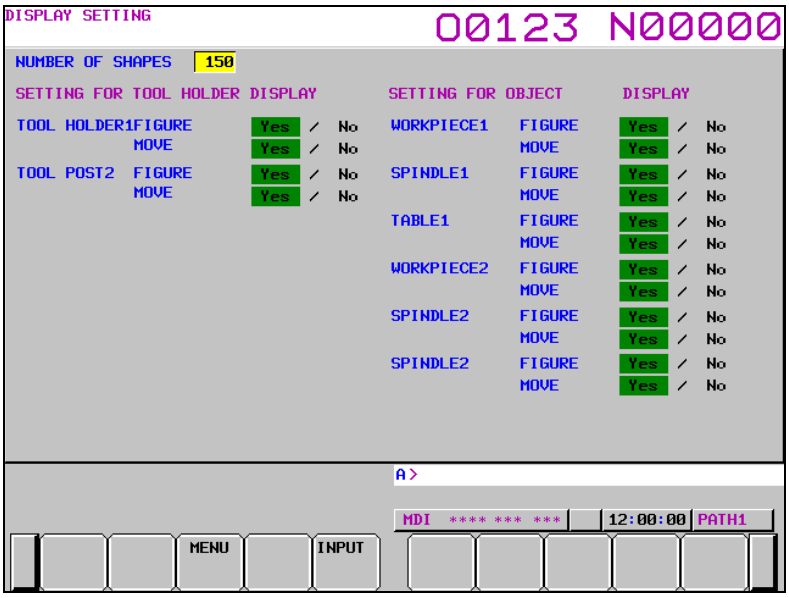


Fig. 12.3.36.15 (a) Display setting screen 10.4”

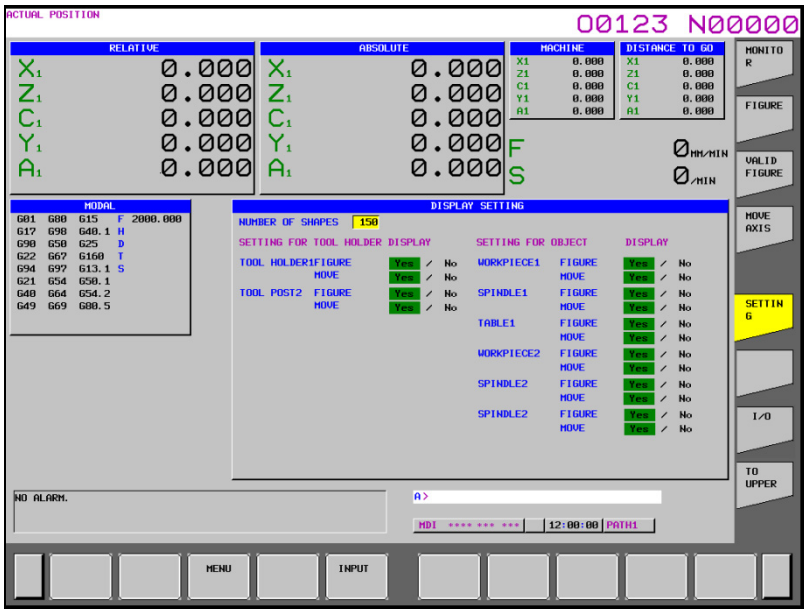


Fig. 12.3.36.15 (b) Display setting screen 15”

When the display of a figure setting screen is set to "NO", the figure setting screen, moving axis setting screen, and interference check valid figure selection screen are not displayed.

12.3.36.16 Drawing coordinate system setting screen

On the drawing coordinate system setting screen, set the drawing coordinate system used to display a figure for check immediately after power-on. The setting is described below.

Table 12.3.36.16 (a) Correspondence between drawing coordinate systems and numbers

Number	Drawing coordinate system
0	Default (T series: Coordinate system YZX/ M series: Coordinate system XYZ)
1	XY
2	ZY
3	YZ
4	XZ

Number	Drawing coordinate system
5	ZX
6	XYZ
7	YXZ
8	YZX

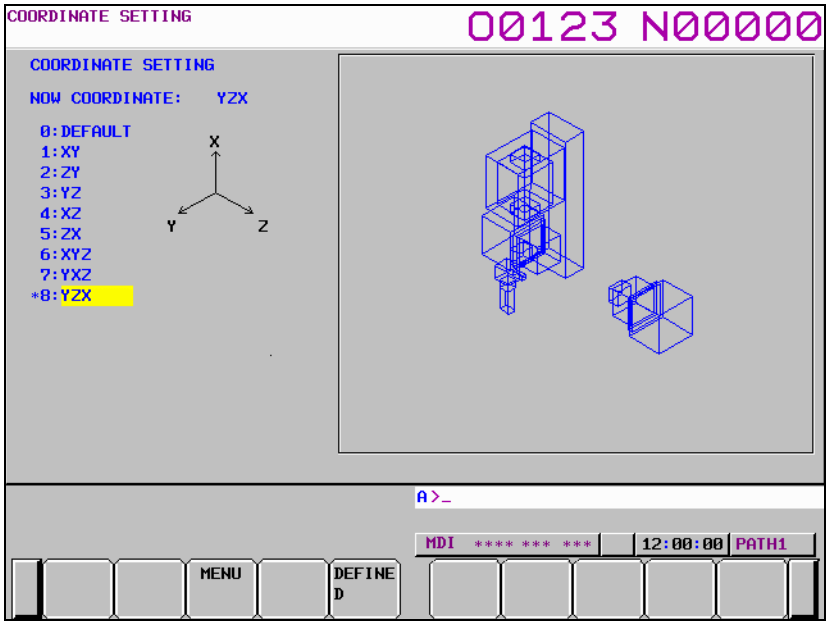


Fig. 12.3.36.16 (a) Drawing coordinate system setting screen 10.4”

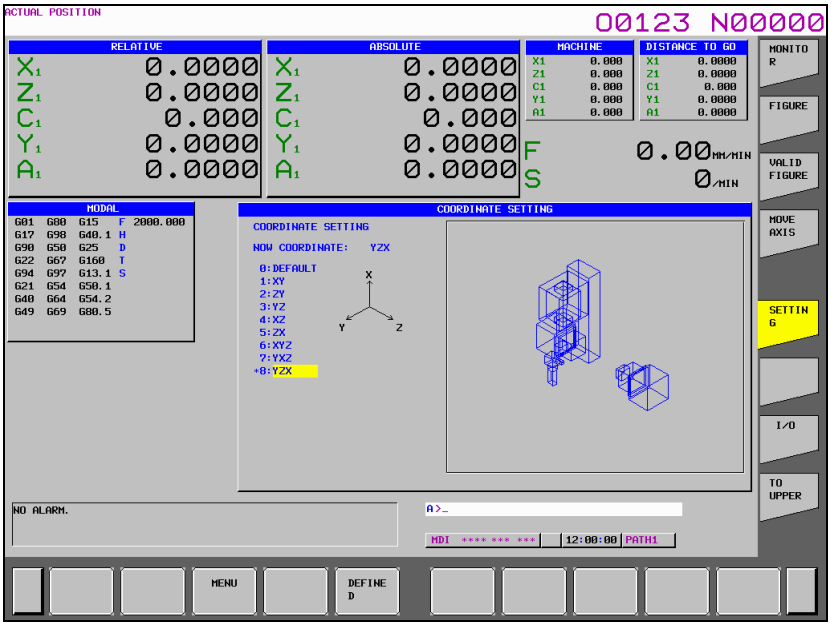


Fig. 12.3.36.16 (b) Drawing coordinate system setting screen 15”

An asterisk (*) is displayed to the left of the currently set item.
The figure of the coordinate system and shape for check are displayed according to the coordinate system at the cursor.

Procedure for setting the drawing coordinate system

To set the drawing coordinate system used to display the shape for check, use the following procedure:

1. Use the cursor keys <↑>, <↓>, <←>, and <→> to move the cursor to the drawing coordinate system you want to set. As the cursor moves, the displayed figure of the coordinate system and figure for check are changed.

2. Press soft key [DEFINED] or <INPUT> key. The selected drawing coordinate system is set.
   At this time, the rotating state of the shape for check on the monitor screen is discarded and the figure returns to the initial position.
   The drawing coordinate system specified on this screen is also used after the restart.

12.3.36.17 Input and output setting data for 3D interference check

All data set on built-in 3D interference check screens can be input and output. The setting data includes the following information items:

- Comment
- Name setting
- Maximum number of displayed shape
- Display setting
- Figure setting
- Shape setting
- Valid figure no.
- Move axis setting

Use the setting I/O screen to input and output data. To display the setting I/O screen, use the following procedure:

1. Press function key .
2. Press soft key [3D INTER.].
3. Press soft key [I/O]. The following screen appears:

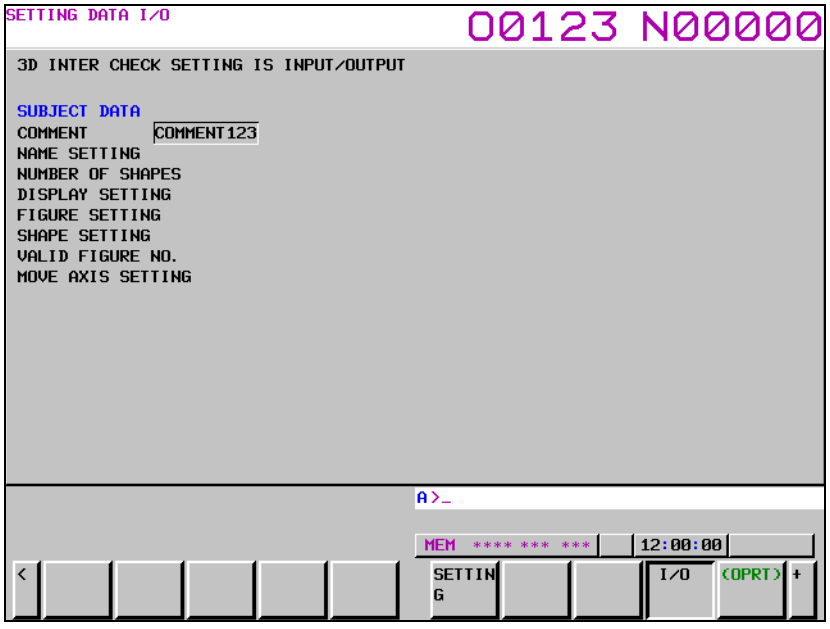


Fig. 12.3.36.17 (a) Setting input/output screen 10.4”

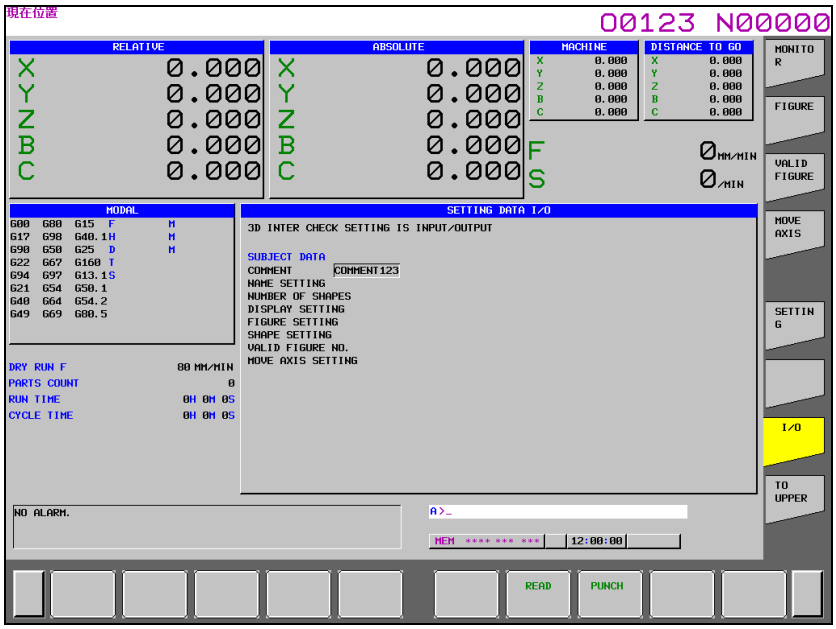


Fig. 12.3.36.17 (b) Setting input/output screen 15”

Input setting data for 3D interference check

The setting data for built-in 3D interference check are loaded into the memory of the CNC from a memory card. The input format is the same as the output format. When the setting data for built-in 3D interference check is loaded which has the corresponding data number as the setting data for built-in 3D interference check already registered in the memory, the loaded data replaces the existing data.

Input setting data for 3D interference check

1. Make sure the input device is ready for reading.
2. Press function key .
3. Press soft key [3D INTER.].
4. Press soft key [+].
5. Press soft key [I/O].
6. Press soft key [(OPRT)].
7. Press the EDIT switch on the machine operator’s panel or enter state emergency stop.
8. Press soft key [F INPUT].
9. Type the name of the file that you want to input.
   If the input file name is omitted, default input file name “3D-INTER.TXT” is assumed.
10. Press the soft key [EXEC].
    This starts reading the setting data for 3D interference check, and “INPUT” blinks in the lower right part of the screen. When the read operation ends, the “INPUT” indication disappears.
    To cancel the input of the program, press the soft key [CAN].

Output setting data for 3D interference check

All 3D interference check data is output from the memory of the CNC to a memory card in the defined output format.

Output setting data for 3D interference check

1. Make sure the output device is ready for output.
2. Press function key .
3. Press soft key [3D INTER.].
4. Press soft key [+].

5. Press soft key [I/O].
6. Press soft key [(OPRT)].
7. Press the EDIT switch on the machine operator's panel or enter state emergency stop.
8. Press soft key [F OUTPUT].
9. Type the file name that you want to output.
   If the file name is omitted, default file name "3D-INTER.TXT" is assumed.
10. Press the soft key [EXEC].
    This starts outputting the setting data for built-in 3D interference check, and "OUTPUT" blinks in the lower right part of the screen. When the read operation ends, the "OUTPUT" indication disappears.
    To cancel the output, press the soft key [CANCEL].

Explanation

- I/O FORMAT

The setting data for built-in 3D interference check is input and output by the following G10 format.

(1) Comment
(Character string) ; The character string enclosed with "(" and ")" is set to the comment.

Example) When "(Ver.1.0.0)" is input, "Ver.1.0.0" is set to the comment.

NOTE

- Up to ten ASCII characters can be set to the comment. When over ten characters are input, ten characters from the head are set to the comment.
- Refer to the Table12.3.36.17 (a) which lists the character that can be set as a comment.
  The character being not listed in the table cannot be used.

Table12.3.36.17 (a)

Character	Explanation
A-Z	English capital letter
a-z	English small letter
0-9	Figure
	Space
"	Quotation mark
#	Hash sign
$	Dollar sign
&	Ampersand
’	Apostrophe
*	Asterisk
+	Plus sign
,	Comma
-	Minus sign
.	Period
/	Slash
:	Colon
;	Semicolon
<	Left angle bracket
=	Equal sign
>	Right angle bracket
?	Question mark
@	At mark
[	Left square bracket
]	Right square bracket
_	Underscore

- When two or more comments are input, the comment at the end is set.
- When the comment is not set to the setting data, the comment is not output.

(2) Name information

G10 L31 P_ Q_ N_ S_ ;

P_ : Type of interference check target item (1 to 3)
1: Object
2: Tool holder
3: Tool

Q_ : Number of the interference check target item
The object (P1): 1 to 6
The tool holder (P2) or the tool (P3): 1 to 4

N_ : Number of the name

S_ : Suffix : 0 to 9

(3) Number of shapes displayed on the shape number list screen
G10 L32 P_ ;
P_ : Number of displayed shapes (0 to 150)

(4) Display setting
G10 L33 P_ Q_ D_ I_ ;
P_ : Type of interference check target item (1 to 2)
1: Object
2: Tool holder
Q_ : Number of the interference check target item
For an object (P1) : 1 to 6
For a tool holder (P2) : 1 to 4
D_ : Figure screen display setting
0 : Displays the figure screen.
1 : Does not display the figure screen.
I_ : Moving axis screen display setting
0 : Displays the moving axis screen.
1 : Does not display the moving axis screen.

(5) Figure information (common to objects and tool holders)
G10 L34 P_ Q_ N_ A_ B_ C_ D_ E_ F_ X_ Y_ Z_ U_ V_ S_ ;
P_ : Type of interference check target item (1 to 2)
1: Object
2: Tool holder
Q_ : Number of the interference check target item
For an object (P1) : 1 to 6
For a tool holder (P2) : 1 to 4
N_ : Figure number
Object (P1) : 1 to 10
Tool holder (P2) : 1 to 120
A_ B_ C_ : Figure element 1 to 6
D_ E_ F_ Shape number
X_ Y_ Z_ : X, Y, and Z coordinates of the reference position
U_ V_ : Reference rotation angles 1 and 2
S_ : Numerical unit
0: Machine unit
1: Input unit

(6) Figure information (for tool holders only)
G10 L35 P2 Q_ N_ X_ Y_ Z_ I_ J_ K_ S_ ;
Q_ : Tool holder number (1 to 4)
N_ : Figure number (1 to 120)
X_ Y_ Z_ : X, Y, and Z coordinates of the tool reference position
I_ J_ K_ : Tool tip direction
(X, Y, and Z vectors from the position of the tool reference position)
S_ : Numerical unit (Note)
0: Machine unit
1: Input unit

NOTE
Numerical unit S can be omitted when data is input, and is ignored when specified. If you want to change the unit of numeric value, specify it in (5), “Figure information (common to objects and tool holders).

(7) Shape definition information

There are the following types of shape definition information: No definition, rectangular parallelepiped, cylinder, and plane, which are classified according to T_.

T0=No definition
T1=Rectangular parallelepiped definition information
T2=Cylinder definition information
T3=Plane definition information

- For no definition (T0)

G10 L36 N_ T0 ;

N_ : Shape number (1 to 150)
T0 : T0 indicates that no data is defined for the shape number (N_).

NOTE
When data is input, definition information for the shape number for which T0 is specified is cleared.

- For rectangular parallelepiped definition information (T1)

G10 L36 N_ T1 X_ Y_ Z_ A_ B_ C_ I_ J_ K_ U_ V_ W_ S_ ;

N_ : Shape number (1 to 150)
T1 : T1 indicates that a rectangular parallelepiped is defined for the shape number (N_).
X_ Y_ Z_ : X, Y, and Z coordinates of the reference vertex
A_ B_ C_ : X, Y, and Z coordinates of adjacent vertex 1
I_ J_ K_ : X, Y, and Z coordinates of adjacent vertex 2
U_ V_ W_ : X ,Y, and Z coordinates of adjacent vertex 3
S_ : Numerical unit
0: Machine unit
1: Input unit

- For cylinder definition information (T2)

G10 L36 N_ T2 X_ Y_ Z_ I_ J_ K_ R_ S_ ;

N_ : Shape number (1 to 150)
T2 : T2 indicates that a cylinder is defined for the shape number (N_).
X_ Y_ Z_ : X, Y, and Z coordinates of the start point
I_ J_ K_ : X, Y, and Z coordinates of the end point
R_ : Radius
S_ : Numerical unit
0: Machine unit
1: Input unit

- For plane definition information (T3)

G10 L36 N_ T3 X_ Y_ Z_ I_ J_ K_ S_;

N_ : Shape number (1 to 150)
T3 : T3 indicates that a plane is defined for the shape number (N_).
X_ Y_ Z_ : X, Y, and Z coordinates of an arbitrary point on the plane
I_ J_ K_ : Normal vectors X, Y, and Z
S_ : Numerical unit
0: Machine unit
1: Input unit

(8) Valid figure number (for objects only)

G10 L37 P1 Q_ N_ ;

Q_ : Object number (1 to 6)

N_ : The valid figure number (0 to 10)
When N0 is specified, the meaning is the following in accordance with parameter ICV (No.10930#3).
0: Figure 1 is effective.
1: No figure. (Removed from interference check target)

(9) Moving axis information

```
G10 L38 P_ Q_ N_ K_
A1(_) D1(_) A2(_) D2(_) A3(_) D3(_)
I1(_) J1(_) X1(_) Y1(_) Z1(_) U1(_) V1(_)
I2(_) J2(_) X2(_) Y2(_) Z2(_) U2(_) V2(_)
M1(_) S1(_) M2(_) S2(_) ・・・ M6(_) S6(_) ;
```

P_	: Type of interference check target item 1: Object 2: Tool holder
Q_	: Number of the interference check target item For an object (P1) : 1 to 6 For a tool holder (P2) : 1 to 4
N_	: Synchronous object number (Note2)
K_	: Absolute path number
A1(_) A2(_) A3(_)	: Axis numbers of linear axes 1, 2, and 3
D1(_) D2(_) D3(_)	: Moving directions of linear axes 1, 2, and 3
I1(_) I2(_)	: Axis numbers of rotation axes 1 and 2
J1(_) J2(_)	: Rotation center axis directions of rotation axes 1 and 2
X1(_) Y1(_) Z1(_) X2(_) Y2(_) Z2(_)	: X, Y, and Z coordinates of the rotation center positions of rotation axes 1 and 2
U1(_) U2(_)	: Inclination angles of rotation axes 1 and 2
V1(_) V2(_)	: Rotation directions of rotation axes 1 and 2
M1(_) M2(_) … M6(_)	: Whether to rotate figure elements 1 to 6 about the master rotation axis (0: Rotated, 1: Not rotated)
S1(_) S2(_) … S6(_)	: Whether to rotate figure elements 1 to 6 about the slave rotation axis (0:Rotate, 1:Not rotate)

NOTE
1 When input, please specify all the addresses except the address that can be omitted.
2 This can be output only when the target is the object (P1). When input, this address can be omitted and if specified, it is ignored.

Example

```
%
G10 L31 P1 Q5 N4 S1 ;
G10 L31 P1 Q5 N4 S2 ;
        :
G10 L32 P51 ;
        :
```

12.3.37 Setting and Displaying Data When the Tool Offset for Milling and Turning Function Is Enabled (15/19-inch Display Unit)

The tool offset for milling and turning function enables offset data to be displayed and operated on a tool offset screen common to milling and turning systems.

Setting and displaying tool offset values (15/19-inch display unit)

Procedure

1 Press function key .

In multi-path control, select the path for which to set tool offset values, using the path selection switch in advance.

2 Press chapter selection vertical soft key [OFFSET].

Alternatively, press function key  several times until the tool offset screen appears.

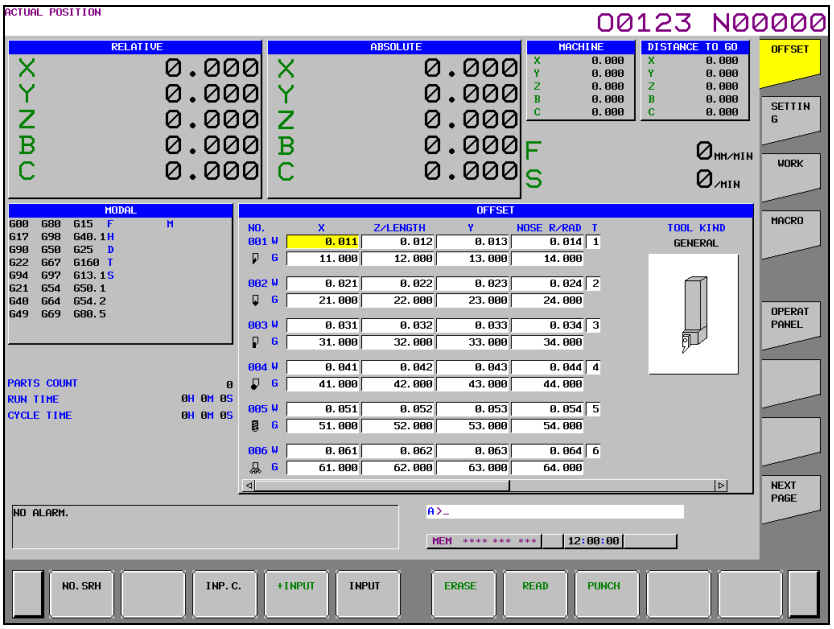


Fig. 12.3.37 (a) Tool offset screen (15-inch display unit)

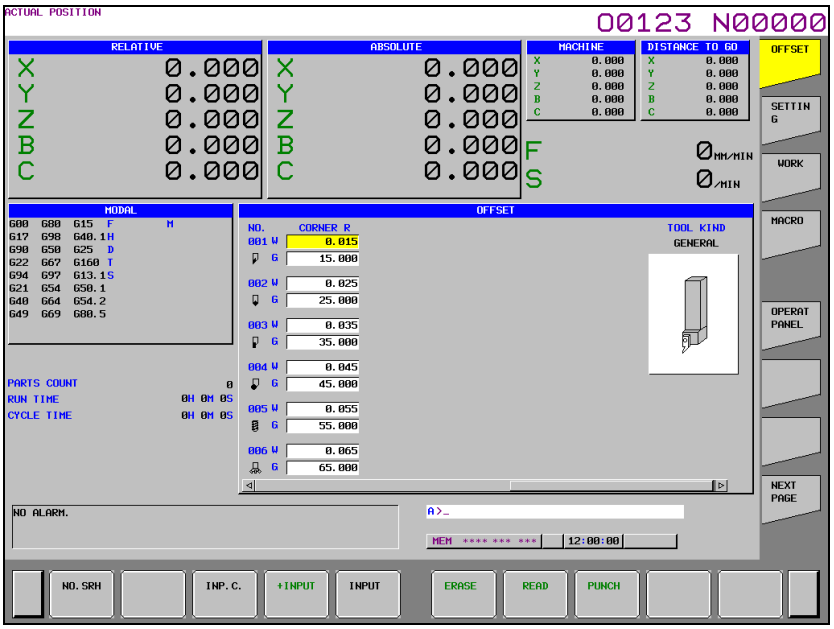


Fig. 12.3.37 (b) Tool offset screen (15-inch display unit)

X : Set the X-axis tool offset value.
Z/LENGTH : Set the Z-axis tool offset value or tool length compensation value.
Y : Set the Y-axis tool offset value.
NOSE R/RAD : Set the tool nose radius compensation value or tool radius compensation value.
T : Set the virtual tool nose direction.
CORNER R : Sets the corner-R offset value.
(This value can be set when the cutting point command option is enabled.)

3 Press page and cursor keys to move the cursor to the offset value you want to set or change.
Alternatively, enter the number of the offset value you want to set or change and press horizontal soft key [NO.SRH].

4 To set the offset value, enter the value and press horizontal soft key [INPUT].
To change the offset value, enter a value to be incremented to or decremented from the current setting and press horizontal soft key [+INPUT].
Alternatively, enter a new offset value and press horizontal soft key [INPUT].

Explanation

- Decimal point programming

The offset value can be entered with a decimal point.

- Another setting method

The tool offset value can be input/output also by using external I/O devices.
See Chapter, “DATA I/O”.
The tool offset value can also be set by using the direct input of offset value measured B, tool offset value counter input, and other functions.

- Number of tool offsets

Sixteen tool offsets are available as standard.
The number of tool offsets can be optionally expanded to 32, 64, or 999.
In a multi-path system, one of the above numbers of tool offsets can be selected for each path.

- Disabling offset value input

By setting bits 0 (WOF) and 1 (GOF) of parameter No. 3290, the input of tool wear compensation values and tool geometry compensation values is disabled.

The input of the tool offset values in any specified range from the MDI can be disabled by setting the number of the first target tool offset value in parameter No. 3294 and the number of target tool offset values starting from the first value in parameter No. 3295.
If offset values are entered continuously, they are handled as follows:

1) If an offset value that can be input is input, then an offset value that cannot be input is input, a warning is output, but the offset value in the input enable range is set.
2) If an offset value that cannot be input is input, then an offset value that can be input is input, a warning is output, and the offset values are not set.

- NOSE R/RAD, T display

When the tool radius/tool nose radius compensation option is not set, NOSE R/RAD, T are not displayed.

12.4 SCREENS DISPLAYED BY FUNCTION KEY 

When the CNC and machine are connected, parameters must be set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor or other parts.
This chapter describes how to set parameters on the MDI unit. Parameters can also be set with external input/output devices such as the memory card (see Chapter, “DATA INPUT/OUTPUT”).
In addition, pitch error compensation data used for improving the precision in positioning with the ball screw on the machine can be set or displayed by the operations under function key .
See Chapter, “ALARM AND SELF-DIAGNOSIS FUNCTIONS” for the diagnosis screens displayed by pressing function key .

Section 12.4, "SCREENS DISPLAYED BY FUNCTION KEY ", consists of the following subsections:

——————— Screens of a 8.4/10.4-inch display unit
12.4.1 Displaying and Setting Parameters 1627
12.4.2 Servo Parameters 1630
12.4.3 Servo Tuning....1631
12.4.4 Spindle Setting....1632
12.4.5 Spindle Tuning....1633
12.4.6 Spindle Monitor 1634
12.4.7 Color Setting Screen 1635
12.4.8 Machining Parameter Tuning....1636
12.4.9 Displaying Memory Data....1644
12.4.10 Parameter Tuning Screen 1646
12.4.11 Periodic Maintenance Screen....1658
12.4.12 System Configuration Screen 1667
12.4.13 Power Consumption Monitoring Screen....1669

——————— Screens of a 15-inch display unit
12.4.14 Displaying and Setting Parameters (15/19-inch Display Unit)....1672
12.4.15 Servo Parameters (15/19-inch Display Unit) 1674
12.4.16 Servo Tuning (15/19-inch Display Unit) 1675
12.4.17 Spindle Setting (15/19-inch Display Unit) 1676
12.4.18 Spindle Tuning (15/19-inch Display Unit) 1677
12.4.19 Spindle Monitor (15/19-inch Display Unit)....1678
12.4.20 Color Setting Screen (15/19-inch Display Unit)....1679
12.4.21 Machining Parameter Tuning (15/19-inch Display Unit) 1680
12.4.22 Displaying Memory Data (15/19-inch Display Unit) 1687
12.4.23 Parameter Tuning Screen (15/19-inch Display Unit)....1690
12.4.24 Periodic Maintenance Screen (15/19-inch Display Unit) 1703
12.4.25 System Configuration Screen (15/19-inch Display Unit) 1710
12.4.26 Power Consumption Monitoring Screen (15/19-inch Display Unit) 1713

Screens of a 8.4/10.4-inch display unit

12.4.1 Displaying and Setting Parameters

When the CNC and machine are connected, parameters are set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor. The setting of parameters depends on the machine. Refer to the parameter list prepared by the machine tool builder.

Normally, the user need not change parameter setting.

Procedure for displaying and setting parameters

Procedure

1 Set 1 for PARAMETER WRITE to enable writing. See the procedure for enabling/disabling parameter writing described below.

2 Press function key 

.

3 Press chapter selection soft key [PARAM] to display the parameter screen.

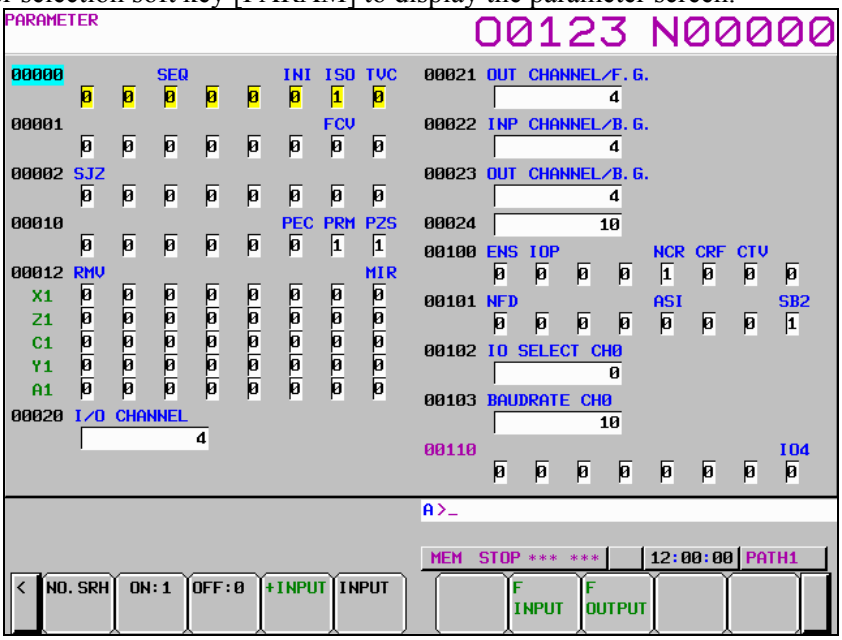


Fig. 12.4.1 (a) PARAMETER screen (10.4-inch display unit)

4 Move the cursor to the parameter number to be set or displayed in either of the following ways:

- Enter the parameter number and press soft key [NO.SRH] .
- Move the cursor to the parameter number using the page keys, 

and 

, and cursor keys, , , , and .

5 To set the parameter, enter a new value with numeric keys and press soft key [INPUT]. The parameter is set to the entered value and the value is displayed.

6 Set 0 for PARAMETER WRITE to disable writing.

Procedure for enabling/displaying parameter writing

Procedure

1 Select the MDI mode or enter state emergency stop.

2 Press function key 

.

3 Press soft key [SETTING] to display the setting screen.

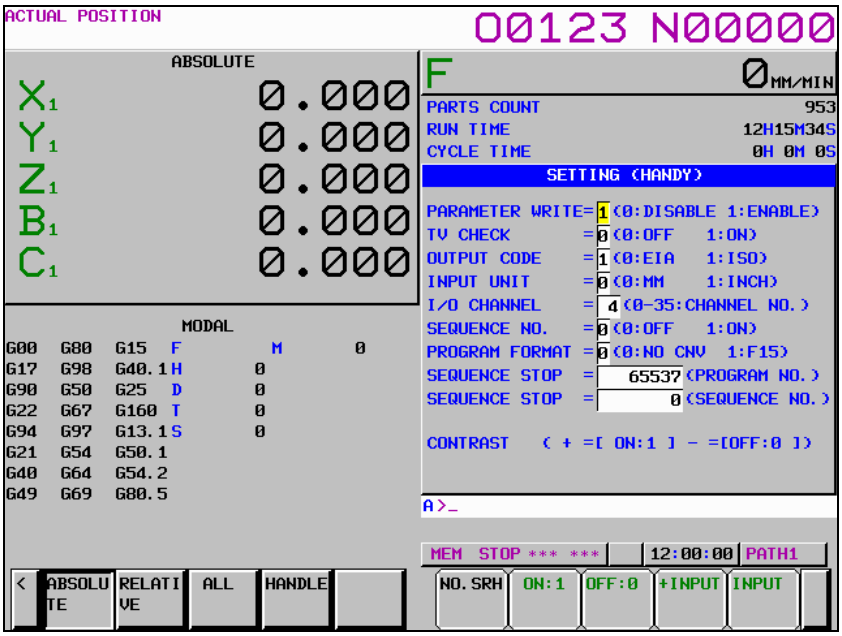


Fig. 12.4.1 (b) SETTING screen (10.4-inch display unit)

4 Move the cursor to PARAMETER WRITE using cursor keys.

5 Press soft key [(OPRT)], then press [ON:1] to enable parameter writing.
At this time, the CNC enters the alarm state SW0100.

6 After setting parameters, return to the setting screen. Move the cursor to PARAMETER WRITE and press soft key [(OPRT)] , then press [OFF:0].

7 Press the RESET key to release the alarm condition.

When an alarm PW0000 occurred, however, it is not released unless the power is turned off and back on.

Explanation

- Setting parameters with external input/output devices

See Chapter, “DATA INPUT/OUTPUT” for setting parameters with external input/output devices such as the memory card.

- Parameters that require turning off the power

Some parameters are not effective until the power is turned off and on again after they are set. Setting such parameters causes alarm PW0000. In this case, turn off the power, then turn it on again.

- Parameter list

Refer to the Parameter Manual (B-64490EN) for the parameter list.

- Setting data

Some parameters can be set on the setting screen if the parameter list indicates "Setting entry is acceptable". Setting 1 for PARAMETER WRITE is not necessary when these parameters are set on the setting screen.

12.4.2 Servo Parameters

This subsection describes the initialization of digital servo parameters performed, for example, at the time of field tuning of the machine tool.

Procedure for servo parameter setting

Procedure

1 Turn on the power in the emergency stop state.
2 Set bit 0 (SVS) of parameter No. 3111 to 1 to display servo setting and tuning screens.
3 Press function key 

, continuous menu key , then soft key [SERVO PARAM].
4 Press soft key [SERVO PARAM] to select the servo parameter screen.
The following screen (Fig. 12.4.2 (a)) appears:

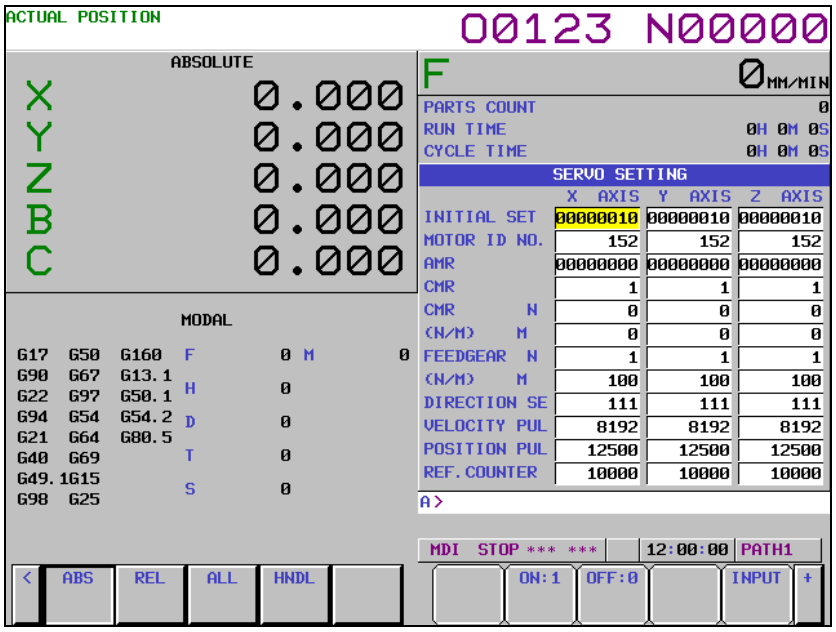


Fig. 12.4.2 (a) Servo parameter setting screen (10.4-inch display unit)

5 With the page keys and cursor keys, move the cursor to the position of data to be set or modified.
6 Key in a desired value then press soft key [INPUT].

12.4.3 Servo Tuning

Data related to servo tuning is displayed and set.

Procedure for servo tuning

Procedure

1 Turn on the power in the emergency stop state.
2 Set bit 0 (SVS) of parameter No. 3111 to 1 to display servo setting and tuning screens.
3 Press function key 

, continuous menu key , then soft key [SERVO PARAM].
4 Press soft key [SERVO TUNE] to select the servo tuning screen.
5 Use the cursor and page keys to enter necessary data for initial setting. The following screen (Fig. 12.4.3 (a)) appears:

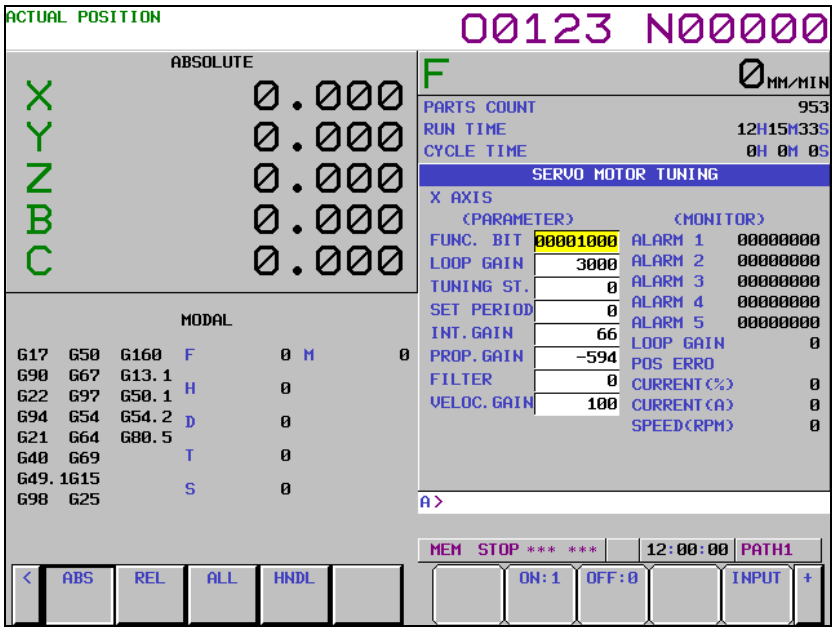


Fig. 12.4.3 (a) Servo tuning screen (10.4-inch display unit)

6 With the page keys and cursor keys, move the cursor to the position of data to be set or modified.
7 Key in a desired value then press soft key [INPUT].

12.4.4 Spindle Setting

Parameters related to spindles are set and displayed. In addition to the parameters, related data can be displayed. Screens for spindle setting, spindle tuning, and spindle monitoring are provided.

Setting spindle parameters

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display spindle setting and tuning screens.

2 Press function key 

, continuous menu key , then soft key [SPINDL SETING].

3 Press soft key [SPINDL SETING] to select the spindle setting screen.
The following screen (Fig. 12.4.4 (a)) appears:

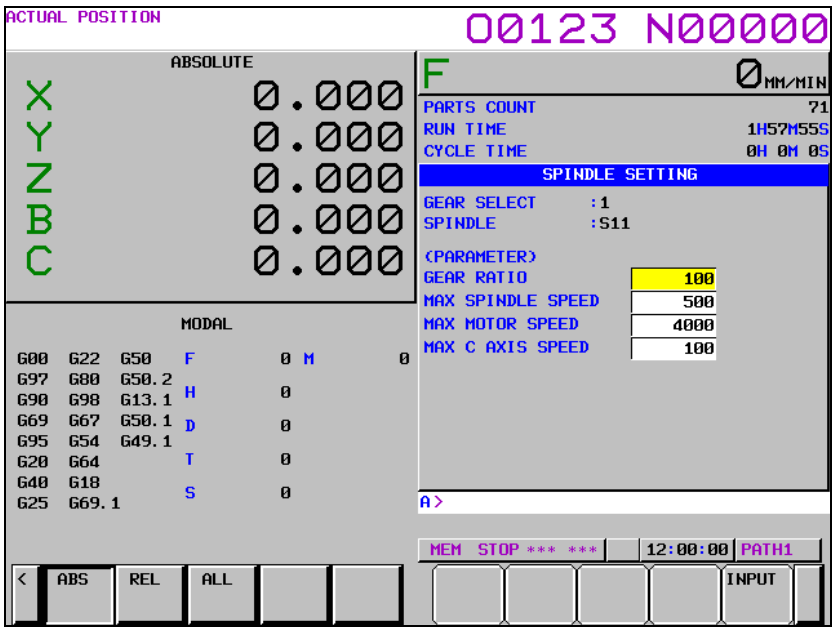


Fig. 12.4.4 (a) Spindle setting screen (10.4-inch display unit)

4 With the page keys and cursor keys, move the cursor to the position of data to be set or modified.

5 Key in a desired value then press soft key [INPUT].

12.4.5 Spindle Tuning

Spindle tuning data is displayed and set.

Setting for spindle tuning

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display spindle setting and tuning screens.
2 Press function key SYSTEM, continuous menu key ▷, then soft key [SPINDL SETING].
3 Press soft key [SPINDL TUNE] to select the spindle tune screen.
4 The following screen (Fig. 12.4.5 (a)) appears:

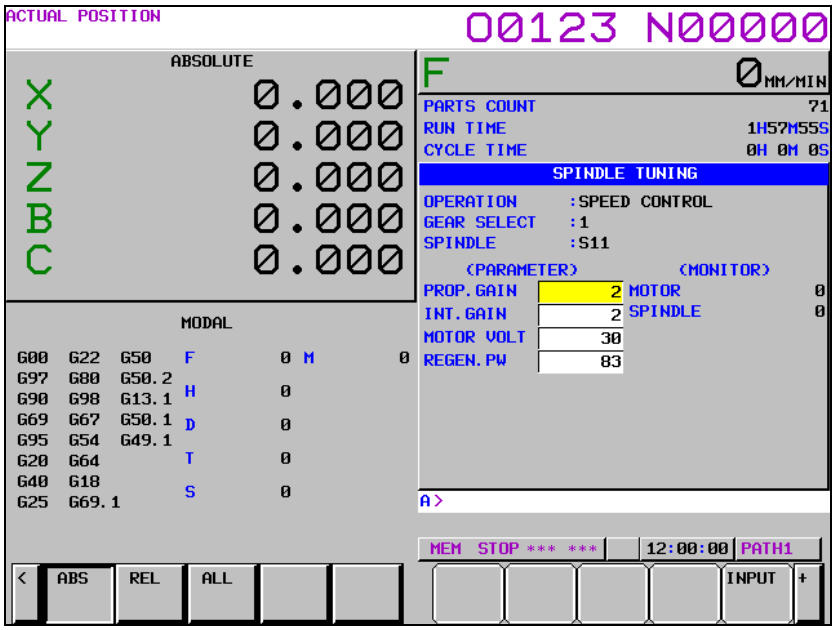


Fig. 12.4.5 (a) Spindle tuning screen (10.4-inch display unit)

5 With the page keys and cursor keys, move the cursor to the position of data to be set or modified.
6 Key in a desired value then press soft key [INPUT].

12.4.6 Spindle Monitor

Spindle-related data is displayed.

Displaying the spindle monitor

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display spindle setting and tuning screens.
2 Press function key SYSTEM, continuous menu key ▷, then soft key [SPINDL SETING].
3 Press soft key [SP.MON] to select the spindle monitor screen.
4 The following screen (Fig. 12.4.6 (a)) appears:

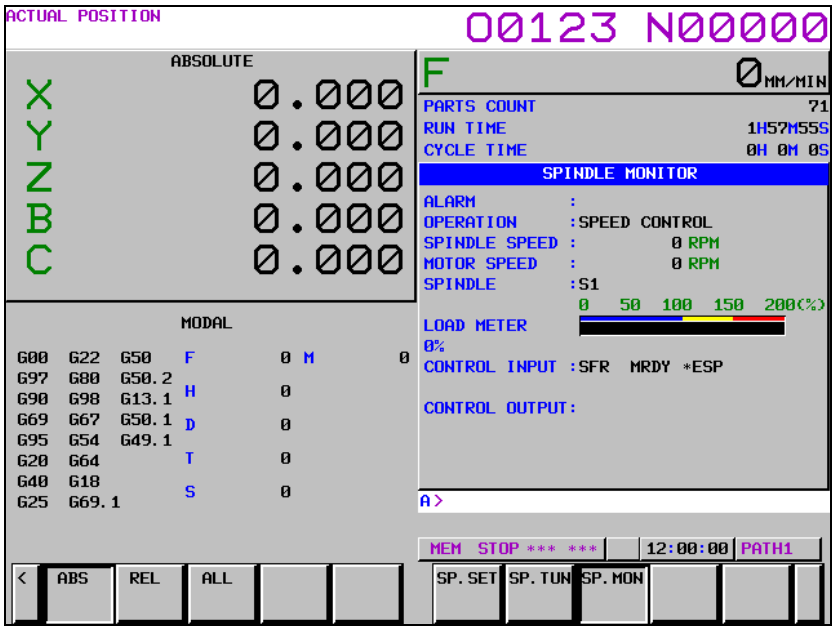


Fig. 12.4.6 (a) Spindle monitor screen (10.4-inch display unit)

12.4.7 Color Setting Screen

Screen colors can be set on the color setting screen.

Displaying the color setting screen

Procedure

1 Press function key SYSTEM.
2 Press the continuous menu key ▷ several times to display soft key [COLOR].
3 Press soft key [COLOR] to display the color setting screen.

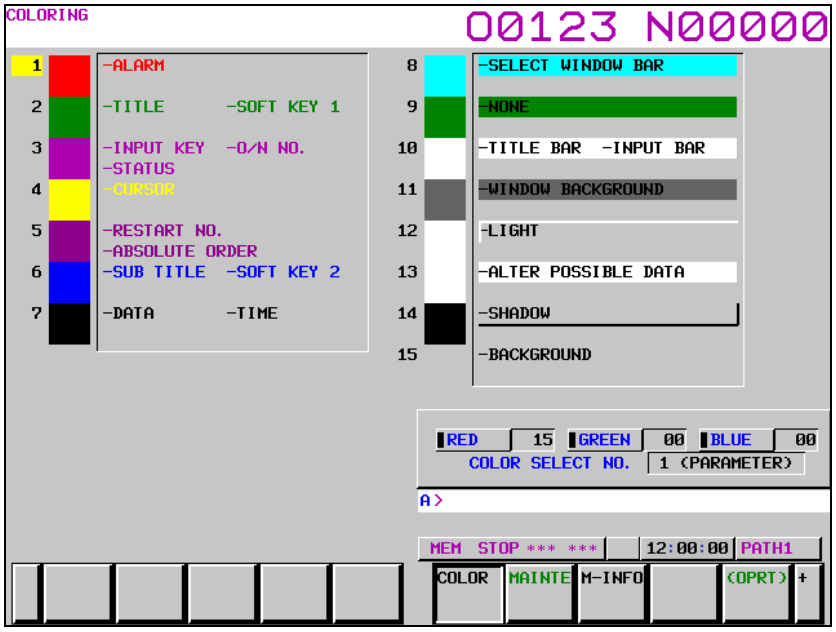


Fig. 12.4.7 (a) Color setting screen (10.4-inch display unit)

Procedure for operating the color setting screen

- Modifying the color (color palette values)

1 Press soft key [(OPRT)]. The soft key display changes to the following operation soft keys:

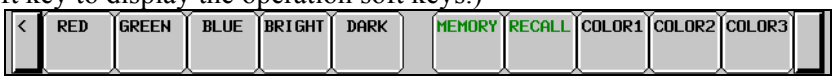


2 Move the cursor to a color number whose color palette values are to be modified.
The current color palette value set for each primary color is displayed.
3 Select a primary color whose setting is to be modified, with the corresponding operation soft key [RED], [GREEN], or [BLUE].
More than one primary color can be selected at the same time.
Each time operation soft key [RED], [GREEN], or [BLUE] is pressed, the operation soft key toggles between selection and deselection.
(When operation soft keys [RED], [GREEN], and [BLUE] are not displayed, press the rightmost soft key to display the operation soft keys.)
4 Select operation soft key [BRIGHT] or [DARK] to modify the brightness of the selected prime color(s).

- Storing the color (color palette values)

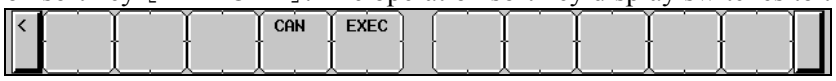
The set color palette values can be stored.
1 Press operation soft key [COLOR1], [COLOR2], or [COLOR3] to select a storage area.

(When operation soft keys [COLOR1], [COLOR2], and [COLOR3] are not displayed, press the rightmost soft key to display the operation soft keys.)



COLOR1 Standard color data parameters Nos. 6581 to 6595
COLOR2 Parameters Nos. 10421 to 10435
COLOR3 Parameters Nos. 10461 to 10475

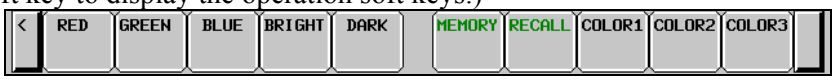
2 Press operation soft key [MEMORY]. The operation soft key display switches to the following:



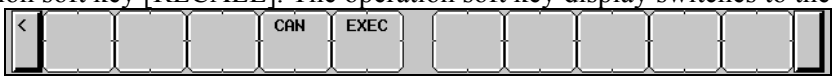
3 Pressing operation soft key [EXEC] stores the current color palette settings in the selected area.
Pressing operation soft key [CAN] or the leftmost key does not store the current color palette settings in the selected area.

- Calling the color (color palette values)

1 Press operation soft key [COLOR1], [COLOR2], or [COLOR3] to select a storage area where color palette values are stored.
(When operation soft keys [COLOR1], [COLOR2], and [COLOR3] are not displayed, press the rightmost soft key to display the operation soft keys.)



2 Press operation soft key [RECALL]. The operation soft key display switches to the following:



3 Pressing operation soft key [EXEC] calls the color palette values from the selected area to enable the color to be modified. This operation is invalid if no color palette values are stored.
Pressing operation soft key [CAN] or the leftmost key does not call the color palette values from the selected area.

NOTE
1 Immediately after the power is turned on, the settings of COLOR1 (parameters) are used for display. If nothing is stored for COLOR1, the default colors are used for display.
2 Do not modify the color setting data parameters directly by MDI key input. When modifying the standard color data, be sure to perform a storage operation on the color setting screen.

12.4.8 Machining Parameter Tuning

12.4.8.1 Machining parameter tuning (AI contour)

In AI contour control, by setting a velocity-emphasized parameter set and precision-emphasized parameter set and setting the precision level matching a machining condition such as rough machining or finish machining on the precision level setting screen or by programming, the parameters suitable for the condition can be automatically calculated to perform machining.
On this screen, the parameter sets for emphasis on velocity (precision level 1) and emphasis on precision (precision level 10) can be set.
Set the following parameters:

- Acceleration rate of acceleration/deceleration before interpolation
- Acceleration change time (bell-shaped)
- Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control

- Allowable acceleration change value for each axis in acceleration change under jerk control in successive linear interpolation operations
- Ratio of the change time of the rate of change of acceleration in smooth bell-shaped acceleration/deceleration before interpolation
- Allowable acceleration rate
- Acceleration rate of acceleration/deceleration after interpolation
- Corner speed difference
- Maximum feedrate
- Items that can be set freely (2 items)

For details of each parameter, see the descriptions of AI contour control and jerk control.
By setting bit 0 (MPR) of parameter No. 13601 to 1, this screen can be hidden.
For the method of setting a precision level, see the description of the precision level selection screen in Subsection 12.3.12, “Precision Level Selection”.

Procedure for machining parameter tuning

Procedure

1 Set the MDI mode.

2 Press function key 

.

3 Press soft key [MCHN TUNING] to display the machining parameter tuning screen.
(If the machining face quality level adjustment function is enabled, press the soft keys [MCHN TUNING] and [AICC] to display the machining parameter tuning screen.)

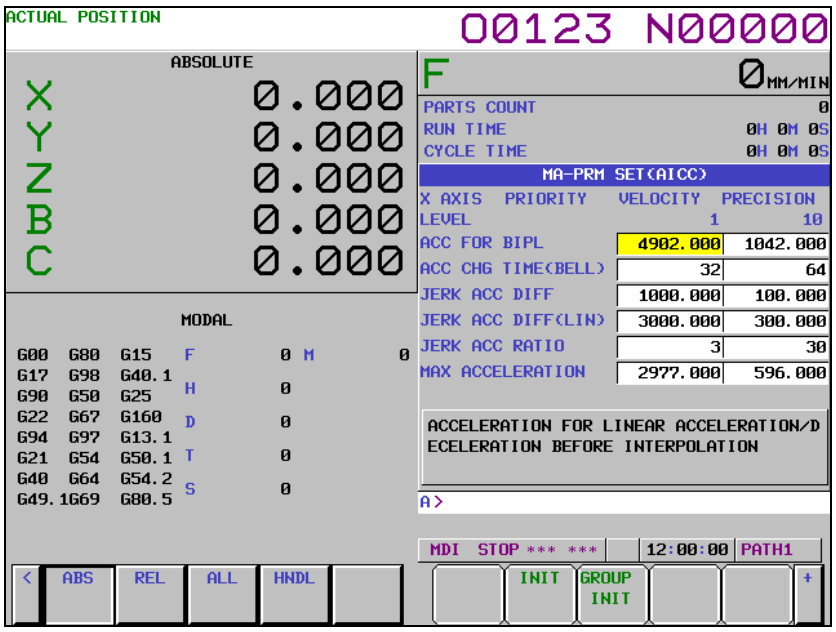


Fig. 12.4.8.1 (a) Machining parameter tuning screen (AI contour) (10.4-inch display unit)

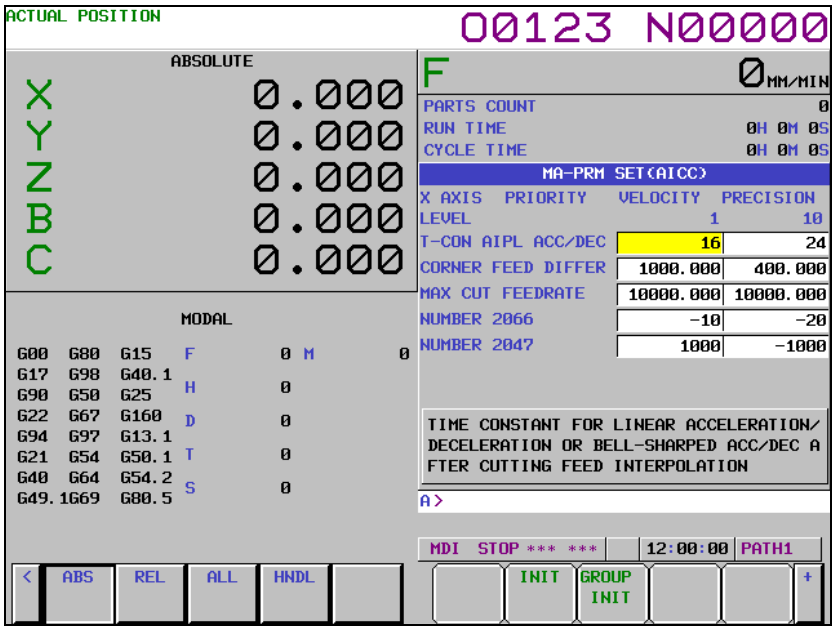


Fig. 12.4.8.1 (b) Machining parameter tuning screen (AI contour) (10.4-inch display unit)

4 Move the cursor to the position of a parameter to be set, as follows:
Press page key PAGE or PAGE, and cursor keys ↑, ↓, ←, and /or → to move the cursor to the parameter.

5 Key in desired data then press the INPUT key on the MDI unit.

6 When data is input, a RMS value is found according to the precision level parameters. (The precision level can be changed on the precision level selection screen or parameter setting screen.) If a RMS value calculation fails, a warning (indicating that automatic setting failed) is displayed.

7 Repeat steps 4 and 5 until all machining parameters are set.

8 In addition to the setting method described above, a parameter setting method using soft keys is available. Pressing soft key [INIT] displays the standard value (recommended by FANUC) of the item selected by the cursor in the key input buffer. Pressing soft key [EXEC] initializes the item to the standard value. Pressing soft key [GROUP INIT] initializes all items of a group (emphasis on velocity or emphasis on precision) selected by the cursor to the standard values.

The Table 12.4.8.1 (a) indicates the initial settings.

Table 12.4.8.1 (a) Initial settings

Setting item	AI contour control		Unit
	Emphasis on velocity (LV1)	Emphasis on precision (LV10)	
Acceleration rate of acceleration/deceleration before interpolation <ACC FOR BIPL>	4902.000	1042.000	mm/sec^2
Acceleration change time (bell-shaped) <ACC CHG TIME(BELL)>	32	64	msec
Allowable acceleration change value <JERK ACC DIFF>	0	0	mm/sec^2
Allowable acceleration change value in successive linear interpolation operations <JERK ACC DIFF(LIN)>	0	0	mm/sec^2
Ratio of the change time of the jerk control <JERK ACC RATIO>	0	0	%
Allowable acceleration rate <MAX AACELERATION>	2977.000	596.000	mm/sec^2
Time constant for acceleration/deceleration after interpolation <T-CON AIPL ACC/DEC>	24	24	msec
Corner speed difference <CORNER FEED DIFFER>	1000	400	mm/min
Maximum cutting speed <MAX CUT FEEDRATE>	10000	10000	mm/min

Explanation

- Look-ahead acceleration/deceleration before interpolation

Set an acceleration rate for a linear portion in look-ahead acceleration/deceleration before interpolation.
Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13610 (velocity-emphasized parameter)
Parameter No. 13611 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1660: Maximum allowable acceleration rate for each axis in acceleration/deceleration before interpolation

- Acceleration change time (bell-shaped)

Set a time constant for a bell-shaped portion in acceleration/ deceleration before look-ahead interpolation.
Unit of data: ms

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13612 (velocity-emphasized parameter)
Parameter No. 13613 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1772: Time constant for bell-shaped look-ahead acceleration/deceleration before interpolation of constant acceleration time type

⚠ CAUTION
A set time constant is applied to all axes. So, a modification made to this item changes the settings for all axes.

- Allowable acceleration change value in velocity control based on acceleration change under jerk control

Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

Set an allowable acceleration change value per ms for each axis in velocity control based on acceleration change under jerk control.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13614 (velocity-emphasized parameter)
Parameter No. 13615 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1788: Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control

NOTE
This setting item is displayed only when the jerk control function is enabled.

- Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations

Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

Set an allowable acceleration change value per ms for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13616 (velocity-emphasized parameter)
Parameter No. 13617 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1789: Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations

⚠ CAUTION
1 For an axis with 0 set in this parameter, the parameters (allowable acceleration change value in velocity control based on acceleration change under jerk control: No. 13614, No.13615) are valid.
2 For an axis with 0 set in the parameter (allowable acceleration change value in velocity control based on acceleration change under jerk control: No. 13614, No.13615), velocity control based on acceleration change is disabled, so that this parameter has no effect.

NOTE
This setting item is displayed only when the jerk control function is enabled.

- Ratio of the change time of the jerk control in smooth bell-shaped acceleration/deceleration before interpolation

Unit of data: %

Set the ratio (in %) of the change time of jerk control to the change time of acceleration in smooth bell-shaped acceleration/deceleration before interpolation.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13618 (velocity-emphasized parameter)
Parameter No. 13619 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1790: Ratio of the change time of the jerk control in smooth bell-shaped acceleration/deceleration before interpolation

NOTE
This setting item is displayed only when the jerk control function is enabled.

- Allowable acceleration rate

Set an allowable acceleration rate in acceleration-based speed determination.
Unit of data: mm/sec^2, $inch/sec^2$, deg/sec^2 (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13620 (velocity-emphasized parameter)
Parameter No. 13621 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1735: Allowable acceleration rate for each axis applicable to the deceleration function based on acceleration in circular interpolation

Parameter No. 1737: Allowable acceleration rate for each axis applicable to the deceleration function based on acceleration in AI contour control

> ⚠ **CAUTION**
> When bit 0 (MCR) of parameter No. 13600 is set to 1, the deceleration function based on acceleration in circular interpolation is not set.

- Time constant for acceleration/deceleration after interpolation

Set a time constant for acceleration/deceleration after interpolation.
Unit of data: ms

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13622 (velocity-emphasized parameter)
Parameter No. 13623 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1769: Time constant for acceleration/deceleration after cutting feed interpolation

- Corner speed difference

Set an allowable corner speed difference used for speed determination.
Unit of data: mm/sec, inch/sec, deg/sec (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13624 (velocity-emphasized parameter)
Parameter No. 13625 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1783: Allowable speed difference for each axis in automatic corner deceleration based on speed difference

- Maximum cutting speed

Set a maximum cutting speed for each axis.
Unit of data: mm/sec, inch/sec, deg/sec (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13626 (velocity-emphasized parameter)
Parameter No. 13627 (precision-emphasized parameter)

Moreover, the following parameter is also set from the precision level:
Parameter No. 1432: Maximum cutting feedrate for each axis in the AI contour control mode

- Arbitrary items

Two arbitrary parameters can be registered. Each item can correspond to a CNC parameter or servo parameter. A parameter number corresponding to each item is to be specified with parameters.

As indicated below (Table 12.4.8 (b)), set the parameters for corresponding parameter numbers, velocity-emphasized parameters (precision level 1), and precision-emphasized parameters (precision level 10).

Table 12.4.8 (b) Parameters related to arbitrary items

	Corresponding parameter number	Setting of velocity-emphasized (precision level 1) value	Setting of precision-emphasized (precision level 10) value
Arbitrary item 1	No.13628	No.13630	No.13632
Arbitrary item 2	No.13629	No.13631	No.13633

- Display
  Tuning target parameter numbers are displayed.

> **⚠ CAUTION**
> As arbitrary items, the numbers of the following parameters cannot be specified:
> - Bit parameter
> - Spindle parameters (Parameters Nos. 4000 to 4799)
> - Real-type parameter
> - Power-off parameter
> - Nonexistent parameter

12.4.8.2 Machining parameter tuning (nano smoothing)

In nano smoothing, by setting a parameter set and setting the smoothing level matching a machining condition on the smoothing level selection screen or by programming, the parameters suitable for the condition can be automatically calculated to perform machining.
On this screen, the parameter sets for emphasis on precision (smoothing level 1) and emphasis on surface smoothing (smoothing level 10) can be set.
Set the following parameters:

- Tolerance
- Tolerance (rotary axis)

For details of each parameter, see the descriptions of nano smoothing.
By setting bit 0 (MPR) of parameter No. 13601 to 1, this screen can be hidden.
For the method of setting a smoothing level, see the description of the smoothing level selection screen in Subsection 12.3.13.1, “Smoothing level selection”.

> **NOTE**
> These setting items are displayed only when machining quality level adjustment is enabled.

Procedure for machining parameter tuning

1 Set the MDI mode.
2 Press function key 

.
3 Press soft key [MCHN TUNING].
4 Press soft key [NANO SMOOTH] to display the machining parameter tuning screen.

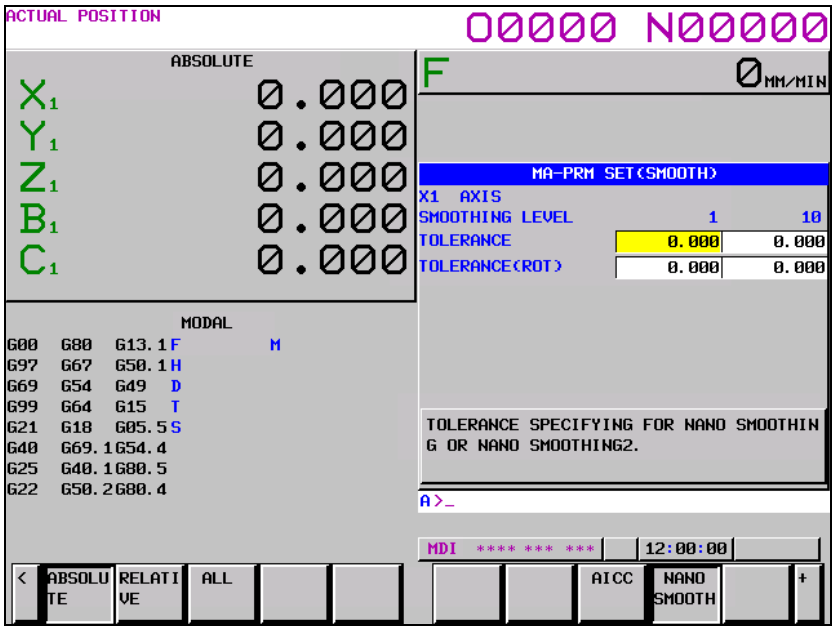


Fig. 12.4.8.2 (a) Machining parameter tuning screen (nano smoothing) (10.4-inch display unit)

5 Move the cursor to the position of a parameter to be set, as follows:
Press page key PAGE ↑ or PAGE ↓, and cursor keys ↑, ↓, ←, and /or →, to move the cursor to the parameter.

6 Key in desired data then press the INPUT key on the MDI unit.

7 When data is input, a RMS value is found according to the smoothing level parameters. (The smoothing level can be changed on the smoothing level selection screen or parameter setting screen.) If a RMS value calculation fails, a warning (indicating that automatic setting failed) is displayed.

8 Repeat steps 5 and 6 until all machining parameters are set.

Explanation

- Tolerance

Set the value specified for the tolerance for nano smoothing.
Unit of data: mm, inch, degree (input unit)

The parameter set on the machining parameter tuning screen (smoothing) is reflected in the following parameters:
Parameter No. 11682 (smoothing level 1)
Parameter No. 11683 (smoothing level 10)

Moreover, the following parameter is also set according to the smoothing level:
Parameter No. 19541: Tolerance specified for nano smoothing

> ⚠ **CAUTION**
> Since the tolerance specified for nano smoothing is common to all axes, changing this item changes the setting for all axes.

- Tolerance (rotary axis)

Set the value specified for the tolerance for the rotary axes in nano smoothing 2.
Unit of data: mm, inch, degree (input unit)

The parameter set on the machining parameter tuning screen (smoothing) is reflected in the following parameters:

Parameter No. 11684 (smoothing level 1)
Parameter No. 11685 (smoothing level 10)

Moreover, the following parameter is also set according to the smoothing level:
Parameter No. 19547: Tolerance specified for rotary axes in nano smoothing 2

> ⚠ **CAUTION**
> This setting item is displayed only when the nano smoothing 2 option is enabled.

12.4.9 Displaying Memory Data

The contents of the CNC memory can be displayed starting at a specified address.

Displaying memory data

Procedure

1 Set bit 0 (MEM) of parameter No. 8950 to 1 to display the memory contents display screen.

2 Press function key [SYSTEM].

3 Press the continuous menu key [▷], then press chapter selection soft key [MEMORY].
The following screen (Fig. 12.4.9 (a)) appears:

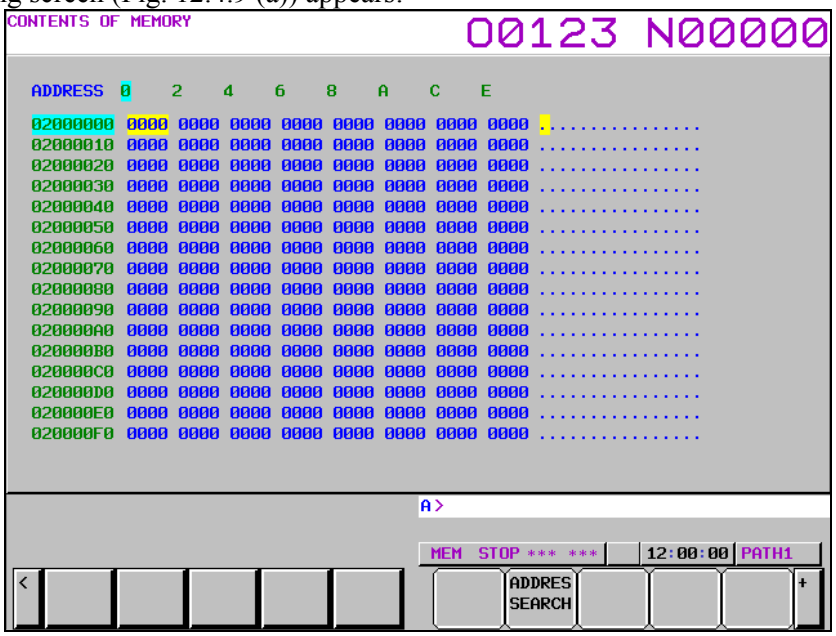


Fig. 12.4.9 (a) Memory contents display screen (10.4-inch display unit)

4 Key in a desired address (hexadecimal) then press soft key [ADDRES SEARCH]. Starting at the specified address, 256-byte data is displayed.
(Example: When you input 100000 then press [ADDRES SEARCH], data is displayed starting at 100000H.)

5 Display data can be switched by using page keys [PAGE↑] and [PAGE↓] and cursor keys [↑], [↓], [←], and/or [→].

6 By pressing soft key [BYTE], [WORD], [LONG], or [DOUBLE], select a data type to be displayed.

Explanation

A memory data display format can be selected from the following four options:

Byte display (1 byte in hexadecimal)
Word display (2 bytes in hexadecimal)
Long display (4 bytes in hexadecimal)
Double display (8 bytes in decimal: Double precision floating-point display)

One screen displays 256-byte memory data.

NOTE
1 When an address is input, "H" for representing "hexadecimal" need not be specified at the end of the address. If H is added, a warning is displayed to indicate that the format is invalid.
2 When word display is selected as the display format, an input address is rounded to a multiple of 2 bytes. When long display or double display is selected, an input address is rounded to a multiple of 4 bytes.

⚠ WARNING
1 If a memory address that must not be accessed in address search is input, a system alarm is issued.
When making an address search, check that the address is accessible and that the address is input correctly.
2 This function is designed for maintenance, and must not be used by general users.

12.4.10 Parameter Tuning Screen

The parameter tuning screen is a screen for parameter setting and tuning designed to achieve the following:

1 The minimum required parameters that must be set when the machine is started up are collectively displayed to facilitate start-up of the machine.
2 The servo tuning screen, spindle tuning screen, and machining parameter tuning screen are displayed for smooth tuning.

The parameter tuning screen consists of a menu screen and several setting screens.

12.4.10.1 Displaying the menu screen and selecting a menu item

The parameter tuning menu screen displays the following items:

[START UP]
- SYSTEM SETTING
- AXIS SETTING
- FSSB (SV AMP)
- FSSB (SP AMP)
- SERVO SETTING
- SPINDLE SETTING
- MISCELLANY

[TUNING]
- SERVO TUNING
- SPINDLE TUNING
- AICC TUNING

On the parameter tuning menu screen, one of the displayed items can be selected to display the corresponding screen. From each setting screen, you can return to the menu screen by performing a soft key operation.

NOTE
1 Some items may not be displayed, depending on the system configuration.
2 When bit 0 (SVS) of parameter No. 3111 is set to 0, "SERVO SETTING" and "SERVO TUNING" are not displayed. When bit 1 (SPS) of parameter No. 3111 is set to 0, "SPINDLE TUNING" is not displayed.

Displaying the menu screen and selecting a setting screen

Procedure

1 Set the MDI mode.
2 Switch the setting of "PARAMETER WRITE" to "ENABLED". For details, see the procedure for "PARAMETER WRITE" in Subsection, “Displaying and Setting Parameters”.
3 Press function key 

.
4 Press the continuous menu key  several times.
5 Press soft key [PARAM TUNING] to display the menu screen for parameter tuning .

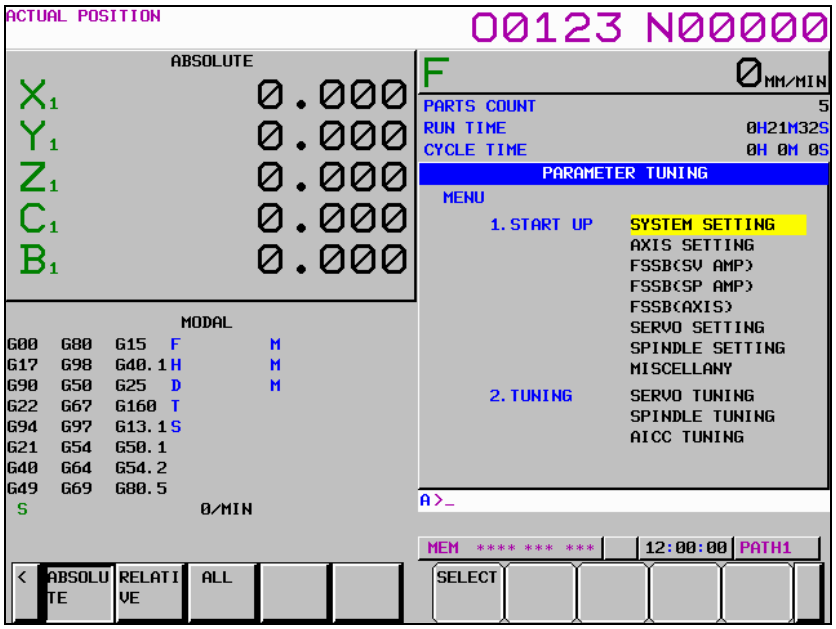


Fig. 12.4.10.1 (a) Menu screen for parameter tuning (10.4-inch display unit)

6 Move the cursor to a desired item by pressing cursor key  or .

7 Press soft key [SELECT]. The screen display switches to the selected screen.

Returning to the menu screen

Procedure

1 Press soft key [SELECT] on the parameter tuning menu screen described in Subsection, “Displaying the menu screen and selecting a menu item”. The screen and soft keys shown below (Fig. 12.4.10.1 (b) are displayed. (The screen below (Fig. 12.4.10.1 (b) is displayed when "AXIS SETTING" is selected.)

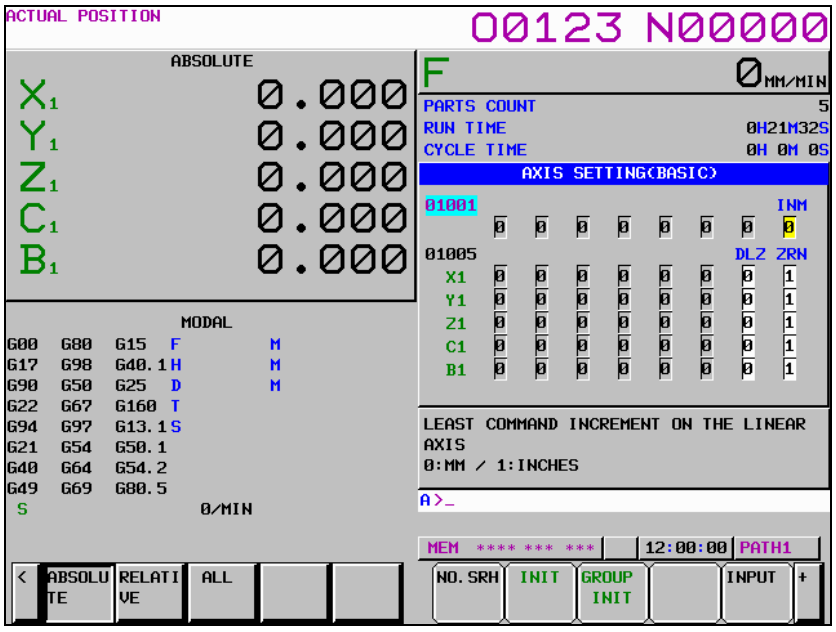


Fig. 12.4.10.1 (b) Axis setting screen (10.4-inch display unit)

2 Press the continuous menu key  several times.

3 Press soft key [MENU].
The screen display returns to the parameter tuning menu screen.
4 Upon completion of parameter setting, switch the setting of "PARAMETER WRITE" to "DISABLED".

NOTE
Some setting screens can also be displayed by a chapter selection soft key. If a screen is selected using a chapter selection soft key, however, you cannot return to the parameter tuning menu screen.

Explanation

- Items displayed with [START UP]

The items of [START UP] indicate the screens for setting the minimum required parameters for starting up the machine.

Table 12.4.10.1 (a) Items displayed with [START UP]

Display item	Description
SYSTEM SETTING	Screen for setting CNC parameters related to the entire system configuration
AXIS SETTING	Screen for setting CNC parameters related to axes, coordinates, feedrate, and acceleration/deceleration
FSSB (SV AMP)	FSSB servo amplifier setting screen
FSSB (SP AMP)	FSSB spindle amplifier setting screen
FSSB (AXIS)	FSSB axis setting screen
SERVO SETTING	Servo setting screen
SPINDLE SETTING	Screen for setting spindle-related parameters
MISCELLANY	Screen for setting parameters related to the allowable number of M code digits and whether to display the servo setting and spindle tuning screens

NOTE
Some items may not be displayed, depending on the system configuration.

- Items displayed with [TUNING]

The items of [TUNING] indicate the screens for servo, spindle, and high-speed high-precision machining tuning.

Fig. 12.4.10.1 (b) Items displayed with [TUNING]

Display item	Description
SERVO TUNING	Servo tuning screen
SPINDLE TUNING	Spindle tuning screen
AICC TUNING	Machining parameter tuning screen

NOTE
1 Some items may not be displayed, depending on the system configuration.
2 When bit 0 (SVS) of parameter No. 3111 is set to 0, "SERVO SETTING" and "SERVO TUNING" are not displayed. When bit 1 (SPS) of parameter No. 3111 is set to 0, "SPINDLE TUNING" is not displayed.

12.4.10.2 Parameter tuning screen (system setting)

This screen enables the parameters related to the entire system configuration to be displayed and modified. The parameters can be initialized to the standard values (recommended by FANUC).

Display and setting

Procedure

1 Move the cursor to [SYSTEM SETTING] by pressing cursor key ↑ or ↓ on the parameter tuning menu screen described in Subsection, “Displaying the menu screen and selecting a menu item”.

2 Press soft key [SELECT]. The screen display switches to the screen and soft keys shown below (Fig. 12.4.10.2 (a)).

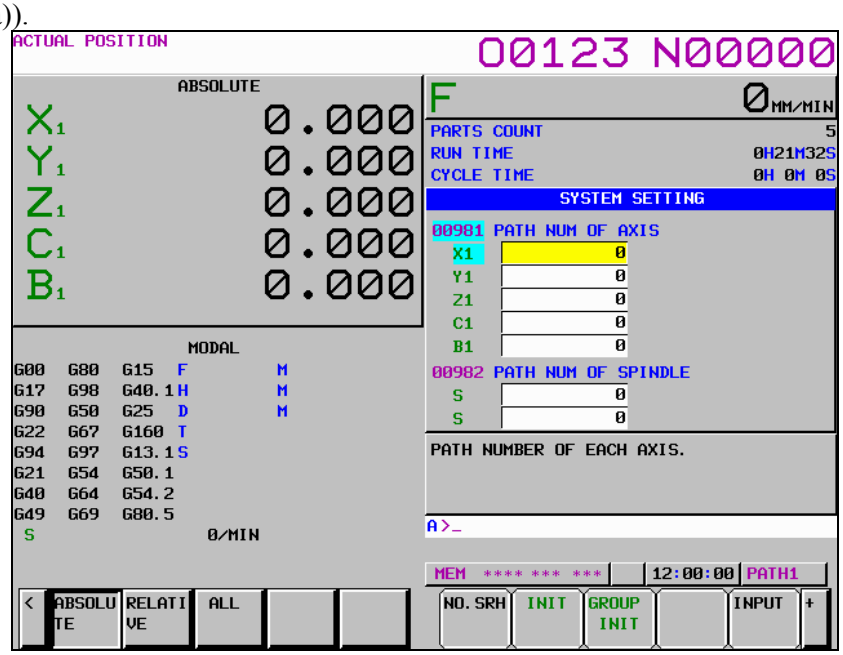


Fig. 12.4.10.2 (a) Parameter tuning screen (system setting) (10.4-inch display unit)

3 Move the cursor to a parameter number to be set or displayed, according to one of the methods below.

- Enter the parameter number and press soft key [NO.SRH] .
- Move the cursor to the desired parameter number by pressing page key PAGE ↑ or PAGE ↓, and cursor keys ↑, ↓, ←, and/or →.

  A brief description of the parameter where the cursor is placed is provided at the bottom of the screen. However, no description is provided when the cursor is placed on multiple bits for bit parameters.

4 Input desired data then press the INPUT key on the MDI unit to set the parameter.

5 Press soft key [INIT]. The standard value (recommended by FANUC) for the item selected by the cursor is displayed in the key input buffer. Pressing soft key [EXEC] in this state initializes the item to the standard value.

6 Press soft key [G_INIT]. A message asking whether to set the group standard values is displayed on the screen. Pressing soft key [EXEC] in this state inputs all of the standard values of the group.

NOTE
1 If the cursor is placed on a parameter that has no standard value assigned, no standard value is input even when [INIT] is pressed.
2 When the cursor is placed on multiple bits for bit parameters, the multiple bits can be input simultaneously. When [INIT] is pressed in this state, the key input buffer displays the standard values for the bits where the cursor is placed. If a bit has no standard value assigned, "*" is displayed for the bit, and no value is input for the bit.
3 When [G_INIT] is pressed, those parameters that have no standard values assigned are not initialized.

12.4.10.3 Parameter tuning screen (axis setting)

This screen enables the CNC parameters related to axes, coordinates, feedrate, and acceleration/deceleration to be displayed and set. The parameters displayed can be divided into four groups:

(Basic) group : The parameters related to basic settings are displayed.
(Coordinate) group : The parameters related to coordinates are displayed.
(Feedrate) group : The parameters related to feedrate are displayed.
(Acceleration/deceleration) group : The parameters related to acceleration/deceleration are displayed.

The parameters can be initialized to the standard values (recommended by FANUC). For the display and setting procedure, see the Subsection, “Parameter tuning screen (system setting)”.

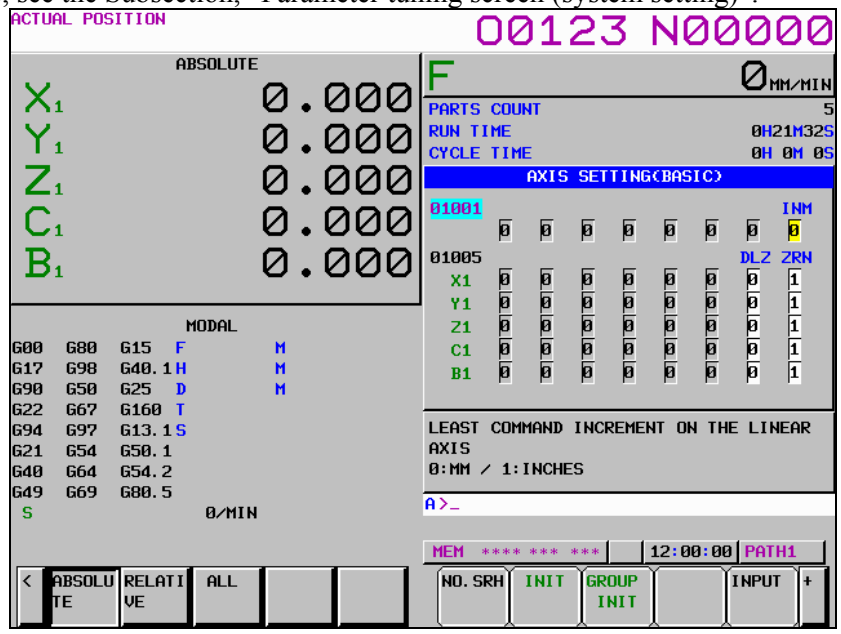


Fig. 12.4.10.3 (a) Parameter tuning screen (axis setting) (10.4-inch display unit)

12.4.10.4 Displaying and setting the FSSB servo amplifier setting screen

From the parameter tuning screen, the FSSB servo amplifier setting screen can be displayed. For details of the FSSB servo amplifier setting screen, see the description of the FSSB servo amplifier setting screen in Subsection, “FSSB Setting” in the Connection Manual (Function) (B-64483EN-1).

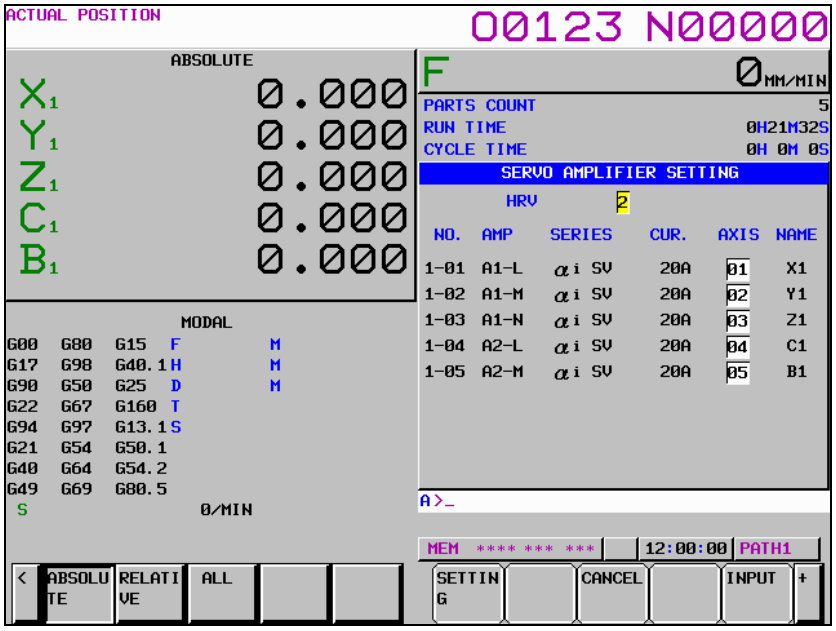


Fig. 12.4.10.4 (a) FSSB servo amplifier setting screen (10.4-inch display unit)

12.4.10.5 Displaying and setting the FSSB spindle amplifier setting screen

From the parameter tuning screen, the FSSB spindle amplifier setting screen can be displayed. For details of the FSSB spindle amplifier setting screen, see the description of the FSSB spindle amplifier setting screen in Subsection, “FSSB Setting” in the Connection Manual (Function) (B-64483EN-1).

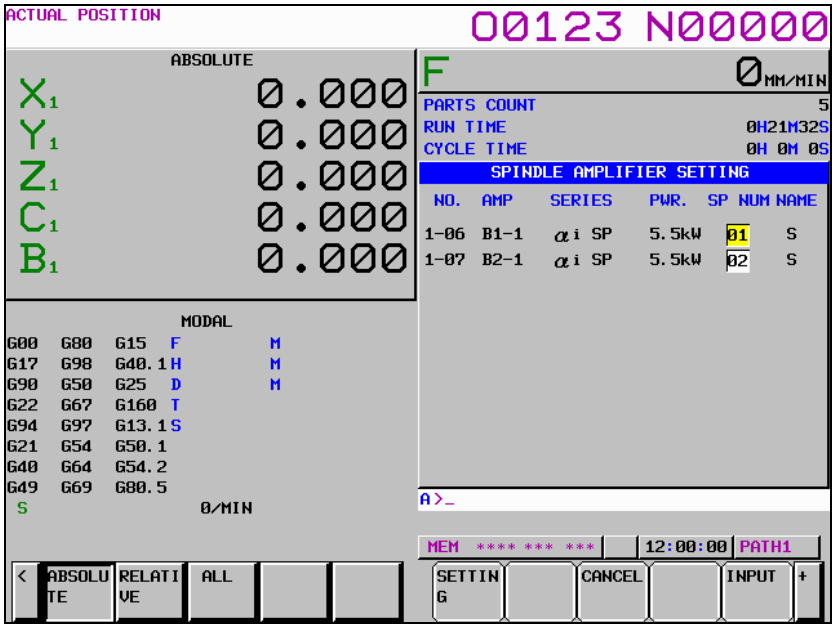


Fig. 12.4.10.5 (a) FSSB spindle amplifier setting screen (10.4-inch display unit)

12.4.10.6 Displaying and setting the FSSB axis setting screen

From the parameter tuning screen, the FSSB axis setting screen can be displayed. For details of the FSSB axis setting screen, see the description of the FSSB axis setting screen in Subsection, “FSSB Setting” in the Connection Manual (Function) (B-64483EN-1).

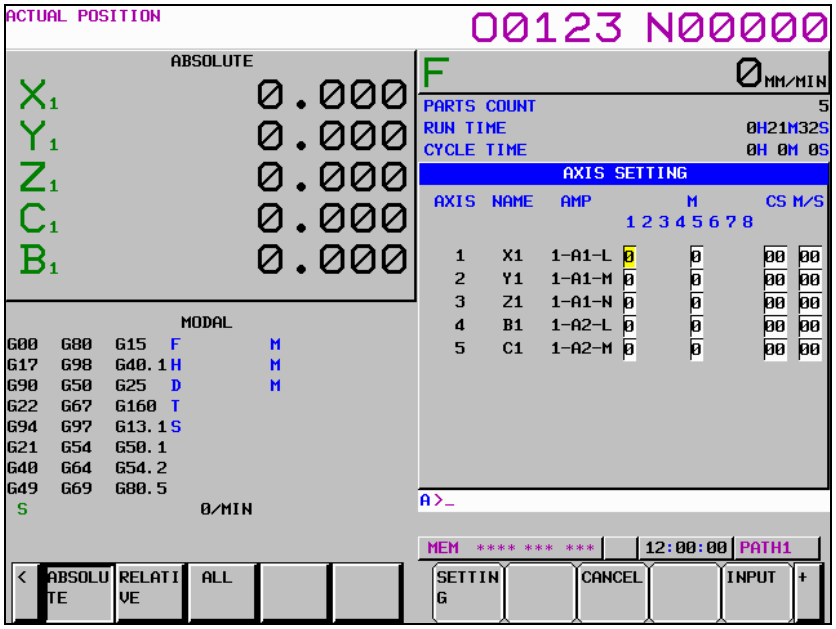


Fig. 12.4.10.6 (a) FSSB axis setting screen (10.4-inch display unit)

12.4.10.7 Displaying and setting the servo setting screen

From the parameter tuning screen, the servo setting screen can be displayed. For details of the servo setting screen, see the Subsection, “Servo Parameters”.

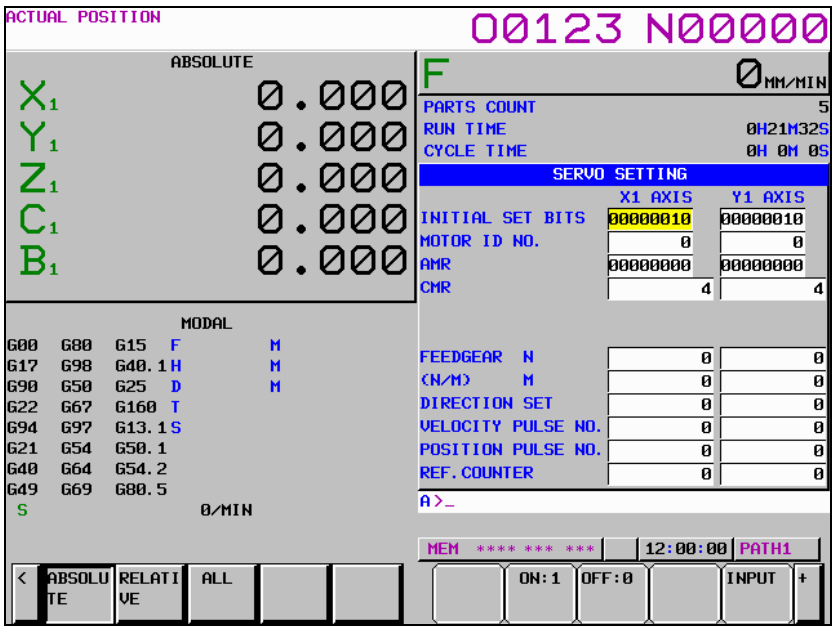


Fig. 12.4.10.7 (a) Servo setting screen (10.4-inch display unit)

12.4.10.8 Parameter tuning screen (spindle setting)

The spindle-related parameters can be displayed and modified.
For the display and setting procedure, see the Subsection, “Parameter tuning screen (system setting)”.

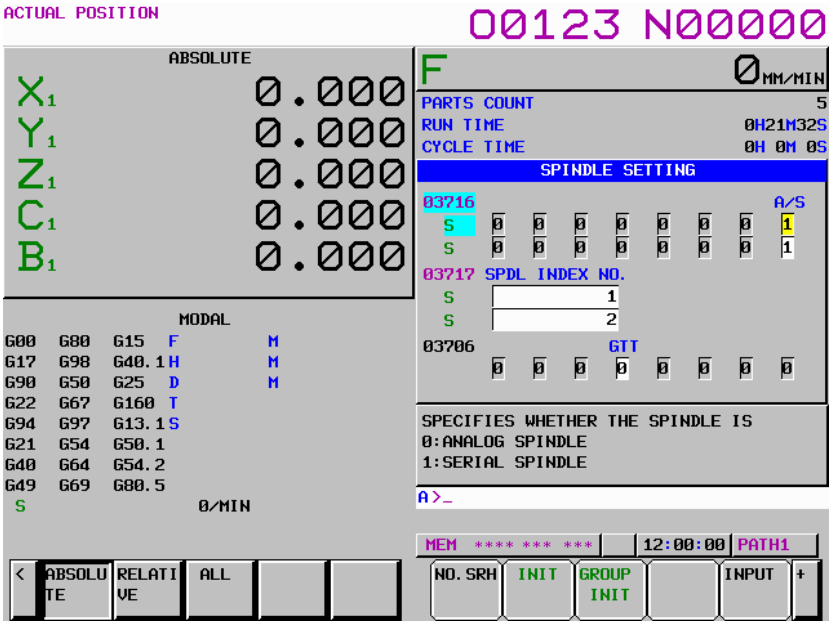


Fig. 12.4.10.8 (a) Parameter tuning screen (spindle setting) (10.4-inch display unit)

12.4.10.9 Parameter tuning screen (miscellaneous settings)

The parameters related to the allowable number of M code digits and whether to display the servo setting and spindle tuning screens can be displayed and modified. Moreover, the parameters can be initialized to the standard values (recommended by FANUC).
For the display and setting procedure, see the Subsection, “Parameter tuning screen (system setting)”.

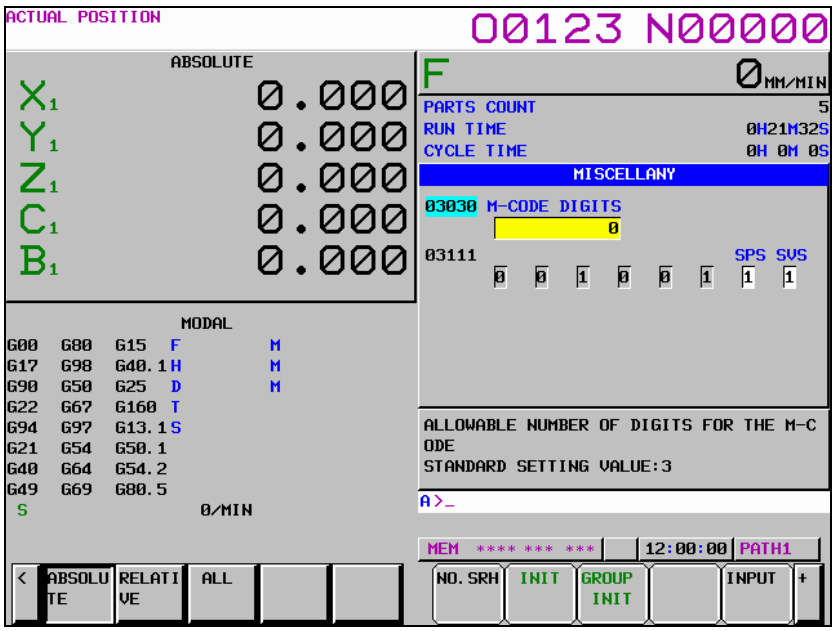


Fig. 12.4.10.9 (a) Parameter tuning screen (Miscellaneous setting) (10.4-inch display unit)

12.4.10.10 Displaying and setting the servo tuning screen

From the parameter tuning screen, the servo tuning screen can be displayed. For details of the servo tuning screen, see the Subsection, “Servo Tuning”.

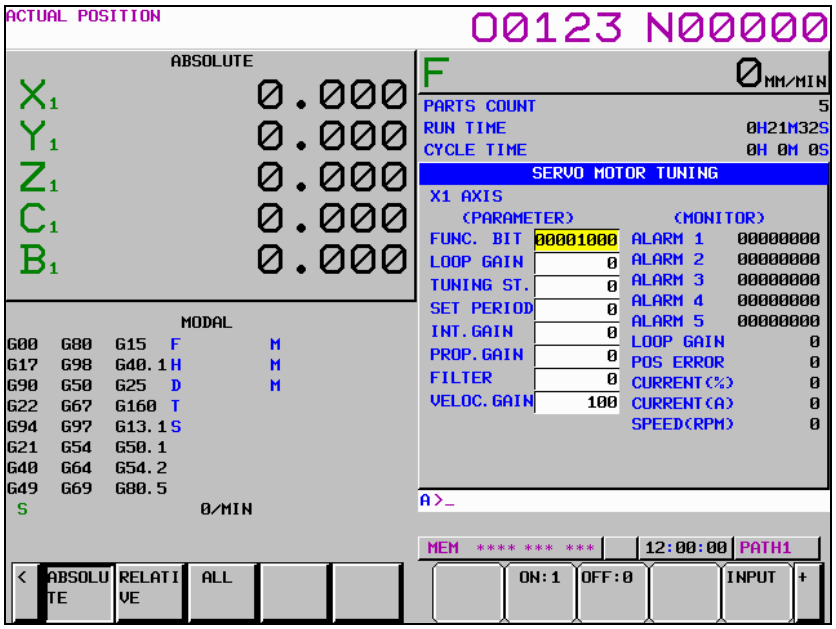


Fig. 12.4.10.10 (a) Servo tuning screen (10.4-inch display unit)

12.4.10.11 Displaying and setting the spindle tuning screen

From the parameter tuning screen, the spindle tuning screen can be displayed. For details of the spindle tuning screen, see the Subsection, “Spindle Tuning”.

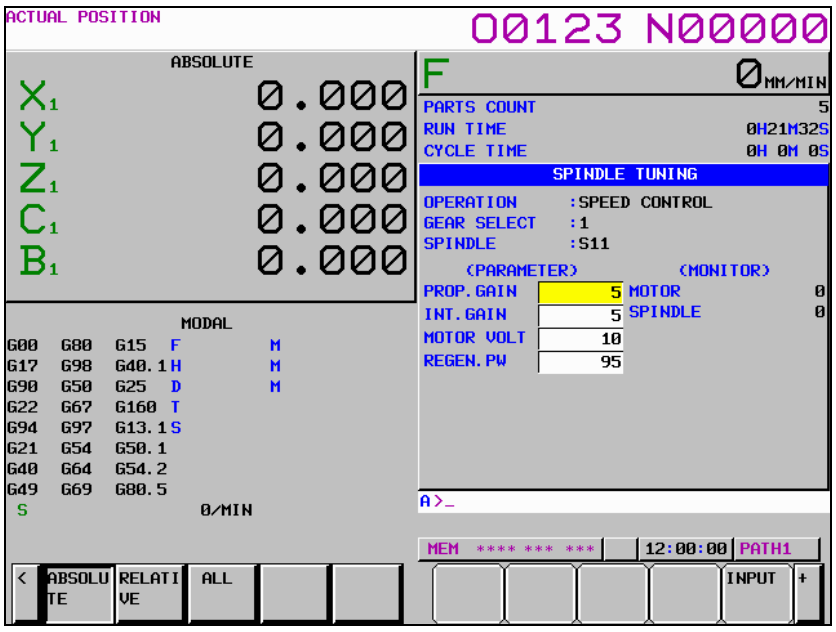


Fig. 12.4.10.11 (a) Spindle tuning screen (10.4-inch display unit)

12.4.10.12 Displaying and setting the machining parameter tuning screen

From the parameter tuning screen, the machining parameter tuning screen can be displayed. For details of the machining parameter tuning screen, see the Subsection, “Machining Parameter Tuning”.

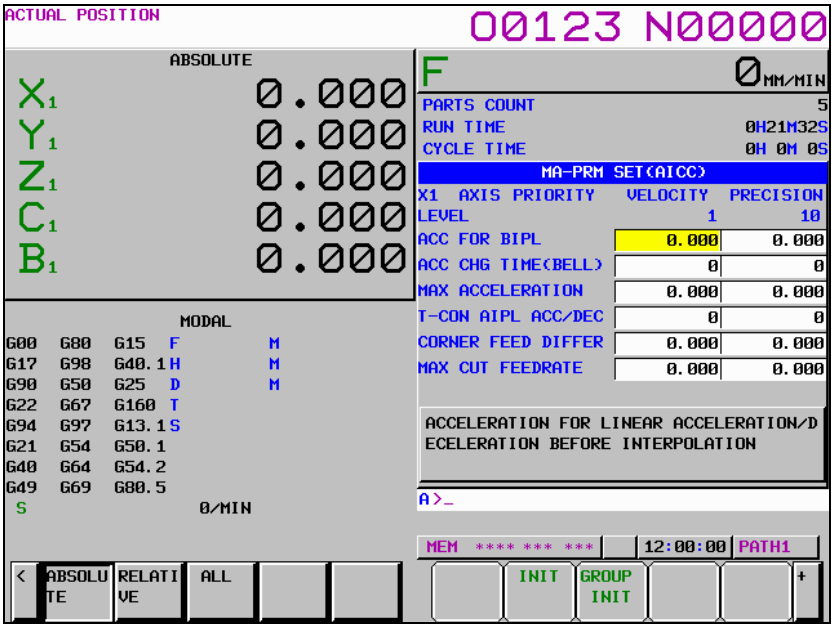


Fig. 12.4.10.12 (a) Machining parameter tuning screen (10.4-inch display unit)

Explanation

- Parameters displayed for parameter tuning

Table 12.4.10 (a) Parameters displayed for parameter tuning (1)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
SYSTEM SETTING	System setting	981		Sets the path of each axis.	
		982		Sets the path of each spindle.	
		983		Sets the T series/M series of each path. 0:T series / 1:M series	
		984#0	LCP	Sets the attribute of each path. 0:Normal / 1:Loader control	*1
		3021		Sets the G/F signal address of each axis.	*2
		3022		Sets the G/F signal address of each spindle.	*3
		3006#0	GDC	The deceleration signal for reference position return is: 0:X signal / 1:G signal	1
		3008#2	XSG	The signal address allocated to address X is: 0:Fixed / 1:Set by the parameter	1
		3013		Address allocated to the deceleration signal for reference position return	*4
		3014		Bit position allocated to the deceleration signal for reference position return	*5

*1 : The value 1 is set for the paths as many as the number of loader paths starting from the greatest path number. For path 1, the value 0 is set at all times.
Example) When the number of loader paths is 3 in a 10-path system:
The value 1 is set for paths 8 to 10. The value 0 is set for others.

*2 : When intra-path axis number ≤ 8, (path number - 1)*10+(intra-path axis number - 1)
When intra-path axis number ≥ 9, no standard value is available.
Example) When path 1 has 9 axes, and path 2 has 3 axes:
0,1,...,7,(none) for axes of path 1, 10,11,12 for axes of path 2

*3 : When intra-path spindle number ≤ 4, (path number - 1)*10+(intra-path spindle number - 1)
When intra-path spindle number ≥ 5, no standard value is available.
Example) When path 1 has 5 spindles, and path 2 has 1 spindle:
0,1,...,4,(none) for spindles of path 1, 10 for spindle of path 2

*4 : When (path number ≤ 3) and (intra-path axis number ≤ 8)
Axes of path 1: 9
Axes of path 2: 7
Axes of path 3: 10
Other axes: No standard value is available.
Example) When path 1 has 9 axes, and path 2 has 3 axes:
9,9,9,9,9,9,9,9,(none) for axes of path 1, 7,7,7 for axes of path 2

*5 : When (path number ≤ 3) and (intra-path axis number ≤ 8)
(Intra-path axis number - 1)
Other axes: No standard value is available.
Example) When path 1 has 9 axes, and path 2 has 3 axes:
0,1,...,7,(none) for axes of path 1, 0,1,2 for axes of path 2

Table 12.4.10 (b) Parameters displayed for parameter tuning (2)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
SPINDLE SETTING	Spindle setting	3716#0	A/S	Sets the type of spindle motor: 0:Analaog / 1:Serial.	
		3717		Sets an amplifier number to be assigned to each spindle.	
		3706#4	GTT	Specifies a spindle gear selection method. 0:M type / 1:T type	
		3718		Sets a suffix to be added to spindle speed display on a screen such as the position display screen.	
		3735		Sets the minimum clamp speed of the spindle motor.	
		3736		Sets the maximum clamp speed of the spindle motor.	
		3741		Sets the maximum spindle speed for gear 1.	
		3742		Sets the maximum spindle speed for gear 2.	
		3743		Sets the maximum spindle speed for gear 3.	
		3744		Sets the maximum spindle speed for gear 4.	
		3772		Sets the maximum speed of the spindle. When 0 is set, the speed is not clamped.	
		4133		Sets the motor model code of the serial spindle. (This setting is not required for an analog spindle.)	
		4019#7	***	The parameters for the serial spindle are: 0:Not automatically set / 1:Automatically set (This setting is not required for an analog spindle.)	

Table 12.4.10 (c) Parameters displayed for parameter tuning (3)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
AXIS SETTING	Basic	1001#0	INM	Least command increment on linear axes: 0:Metric (millimeter machine) / 1:Inch (inch machine)	
		1005#0	ZRNx	When an automatic operation (other than G28) is executed before reference position return: 0:Alarm is issued (PS224) / 1:No alarm is issued.	0
		1005#1	DLZx	Reference position return without dogs: 0:Disabled / 1:Enabled	
		1006#0	ROTx	Setting of linear axes or rotary axis: 0:Linear axis / 1:Rotation axis	
		1006#3	DIAx	Setting of the amount of travel: 0:Radius specification / 1: Diameter specification	
		1006#5	ZMIx	Reference position return direction: 0:Plus direction / 1:Minus direction	
		1008#0	ROAx	Rotation axis roll-over function: 0:Disabled / 1:Enabled	1
		1008#2	RRLx	With the amount of travel per revolution, relative coordinates are: 0:Not rounded / 1:Rounded	1
		1013#0	ISAx	Sets the least input increment and least command increment: 0:IS-B / 1:IS-A	
		1013#1	ISCx	Sets the least input increment and least command increment: 0:IS-B / 1:IS-C	
		1013#2	ISDx	Sets the least input increment and least command increment: 0:IS-B / 1:IS-D	
		1013#3	ISEx	Sets the least input increment and least command increment: 0:IS-B / 1:IS-E	
		1020		Program axis name	*1
		1022		Sets each axis in the basic coordinate system.	*2
		1023		Servo axis number	
		1815#1	OPTx	A separate pulse coder is: 0:Not used / 1:Used	
		1815#4	APZx	The correspondence between machine positions and absolute-position detector positions is: 0:Not established / 1:Established	
		1815#5	APCx	The position detector used is: 0:Other than an absolute-position detector / 1:Absolute-position detector	
		1825		Servo loop gain	
		1826		Effective area	
		1828		Positional deviation limit during travel	
		1829		Positional deviation limit during stop	500

*1 : For the M series: 88(X),89(Y),90(Z) sequentially starting with the first axis (no standard value for the fourth axis and up)
For the T series: 88(X),90(Z) (no standard value for the third axis and up)

*2 : For the M series: 1,2,3 sequentially starting with the first axis (no standard value for the fourth axis and up)
For the T series: 1,3 (no standard value for the third axis and up)

Table 12.4.10 (d) Parameters displayed for parameter tuning (4)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
AXIS SETTING	Coordinate	1240		Machine coordinate of the first reference position	
		1241		Machine coordinate of the second reference position	
		1260		Amount of travel per revolution of a rotary axis	360.000
		1320		Coordinate of the boundary of stored stroke check 1 in the positive direction	
		1321		Coordinate of the boundary of stored stroke check 1 in the negative direction	
	Feedrate	1401#6	RDR	For a rapid traverse command, dry run is: 0:Disabled / 1:Enabled	0
		1410		Dry run feedrate	
		1420		Rapid traverse rate	
		1421		Rapid traverse override F0 rate	
		1423		Jog feedrate	
		1424		Manual rapid traverse rate	
		1425		FL feedrate for reference position return	
		1428		Reference position return feedrate	
		1430		Maximum cutting feedrate	
	Acceleration/deceleration	1610#0	CTLx	Acceleration/deceleration in cutting feed or dry run during cutting feed: 0: Exponential acceleration/deceleration is applied / 1: Linear acceleration/deceleration after interpolation is applied	
		1610#4	JGLx	Acceleration/deceleration for jog feed is: 0: Exponential acceleration/deceleration / 1: Same as acceleration/deceleration for cutting feed (The settings of bit 1 (CTBx) and bit 0 (CTLx) of parameter No. 1610 are followed.)	
		1620		Time constant for linear acceleration/deceleration for rapid traverse	
		1622		Time constant for acceleration/deceleration for cutting feed	
		1623		FL feedrate for acceleration/deceleration after interpolation for cutting feed	
		1624		Time constant for acceleration/deceleration for jog feed	
		1625		FL feedrate for exponential acceleration/deceleration for jog feed	
MISCELLANY	MISC	3030		Allowable number of digits of an M code	3
		3111#0	SVS	The servo setting screen is: 0:Not displayed / 1:Displayed	1
		3111#1	SPS	The spindle setting screen is: 0:Not displayed / 1:Displayed	1

12.4.11 Periodic Maintenance Screen

Periodic maintenance screens are used for managing consumables (such as the backlight of a LCD unit and backup batteries). By setting the name of a consumable, its life time, and the method for counting down the life time, the life of the consumable can be counted down in the appropriate method set for the consumable, and the remaining life time can be displayed.
With these screens, the user can easily manage consumables that require periodic replacement.
The total number of item that can be managed is 10 in all path.

Explanation

There are four periodic maintenance screens: the status screen, the setting screen, the machine menu screen, and the NC menu screen.

Status screen : Item names, remaining times, and count statuses are displayed, and item names are set.
Setting screen : Life times, remaining times, count types (count-down method), and path number (in case of multi-path system) are set.
Machine menu screen : The names of consumables in the machine can be registered.
NC menu screen : The names of consumables in the NC are already registered.

Using periodic maintenance screens

<1> Referencing a periodic maintenance screen
Display the status screen. For the meanings of items on the status screen, see Item, "**Status screen**".
<2> Adding the name of a new consumable item to a periodic maintenance screen or editing an existing consumable item on the screen
The name of a consumable item can be added or edited on the machine menu screen. For details, see Item, "**Machine menu screen**".
<3> Adding or editing a life time, remaining time, and a method of counting down the remaining time for a consumable on a periodic maintenance screen
A life time and remaining time can be added or edited on the setting screen. For details, see Item, "**Setting screen**".
<4> Newly displaying the item name and remaining time of a consumable on a periodic maintenance screen.
1 Setting an item name
Select the item name of a consumable to be displayed from the machine menu screen or NC menu screen, or enter the name using the MDI keys. For the procedure, see Item, "**Item name**" in "**Status screen**".
2 Setting a life time, remaining time, and count type
Select the life time, remaining time, and count type of a consumable to be displayed from the setting screen. For the procedure, see Item, "**Remaining time**" in "**Status screen**".

Procedure for displaying a periodic maintenance screen

1 Press function key .
2 Press the continuous menu key  several times to display [PERIOD MAINTE].
3 Press soft key [PERIOD MAINTE] to display the periodic maintenance screen.

Status screen

When soft key [STATUS] is pressed, the status screen is displayed. The status screen shows the item names, count statuses, and remaining times of managed consumables.

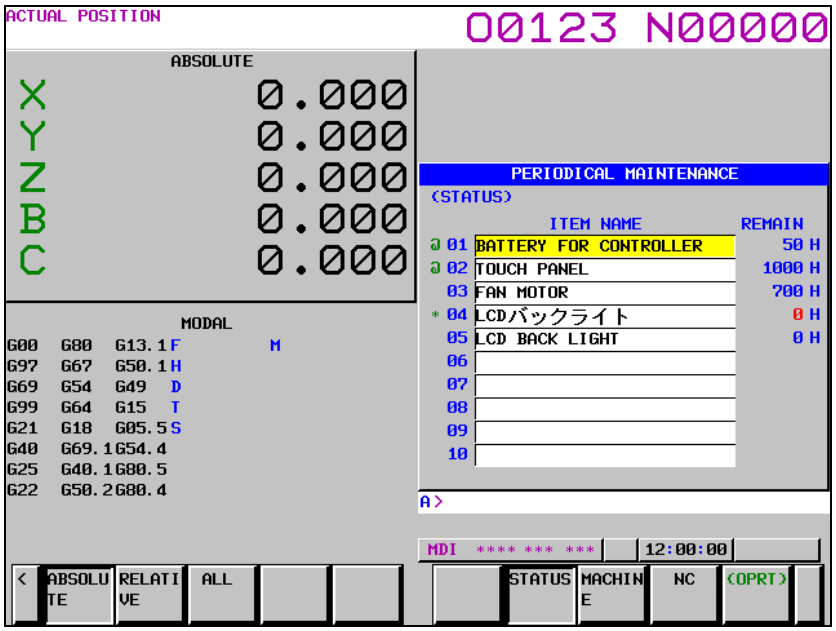


Fig. 12.4.11 (a) Status screen

- Item name

As the item name, set the name of a consumable to be managed by periodic maintenance. To set an item name, select a name from the machine menu screen or NC menu screen, or directly enter the name using the MDI keys.

Setting an item name from a menu screen

1 On the status screen, move the cursor to a target item name, press soft key [(OPRT)], then press soft key [ENTRY].
2 Press soft key [MACHINE] or [NC] to display the machine menu screen or NC menu screen.
3 Move the cursor to an existing item name on the menu screen, press soft key [(OPRT)], and press soft key [SELECT] then [EXEC] successively.
4 The screen display returns to the status screen, and the item name selected on the menu screen is added to the status screen.

Initially, there is no item name set on the machine menu screen, so item names must be registered in advance. For the registration method, see the description of the procedure for registering item names on the machine menu screen.

Setting an item name using the MDI keys

1 Press soft key [(OPRT)].
2 Type the alphanumeric characters to be input, then press soft key [INPUT].
3 The entered item name is registered on the status screen.

When soft key [+INPUT] is pressed instead of soft key [INPUT], the entered characters can be added to an existing item name.
When typing 2-byte characters, type "*" before and after the character codes. The 2-byte character codes must conform to FANUC codes. (See Appendix, "FANUC 2-BYTE CHARACTER CODE TABLE".)
An item name to be registered must be up to 24 characters long if it consists of alphanumeric characters only; or it must be up to 12 characters long if it consists of 2-byte characters only.
Example: To register "LCD バックライト", enter the following:

>LCD*110E10F410CC114010B610FE*_

NOTE
1 An asterisk "*" is used as a control code, so it cannot be used in item names. In addition, characters "[", "]", "(", and ")" cannot be used in item names.
2 When an item name consisting of both alphanumeric characters and 2-byte characters is registered, the warning "DATA OUT OF RANGE" may be output.

Deleting an item name
To delete a registered item name, move the cursor to the item name, press soft key [ERASE], then press soft key [EXEC].
When an item name is deleted, its life time, remaining time, and count type are deleted at the same time.

- Remaining time

As the remaining time, the period of time left until the time for replacement is reached by count down operation is displayed.
If the percentage of the remaining time to the life time has reached to the value (%) specified by parameter No. 8911, or smaller, the remaining time is displayed in red.
Even after the life time has expired, count down operation continues.

Setting the remaining time
1 On the status screen, place the cursor on an item for which the remaining time is to be set (the item name must have been set in advance).
2 Press soft key [(OPRT)], then press soft key [CHANGE].
3 The screen display changes to the setting screen.
4 Set the life time, remaining time, and count type. For the setting method and other information, see Item, "**Setting screen**".

NOTE
On the status screen, the remaining time and life time cannot be set.
These items must be set on the setting screen.

- Count status

The count status is indicated on the left side of the item number as follows:

Indication	Count status
Blank	Counting stopped
@	Counting in progress
*	Life expired

Setting screen

On the setting screen, the life time, remaining time, and count type of a managed consumable are set.

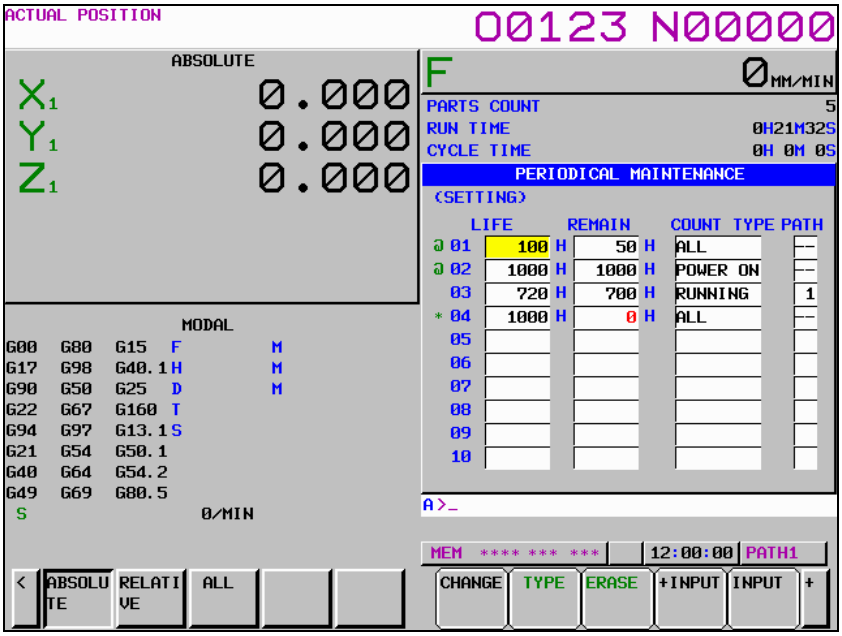


Fig. 12.4.11 (b) Setting screen

Display procedure

1 When the status screen is displayed, press soft key [(OPRT)].
2 Press soft key [CHANGE].

- Life time

Set the life time of a consumable.

Move the cursor to an existing item, type a life time, then press soft key [INPUT] (or the 

key). The life time is then set, and the same value is set also as the remaining time.
At this time, "------" is indicated in the count type field.
When soft key [+INPUT] is pressed, the entered value can be added to the life time already set. The same value as the added value is added also to the remaining time.
A value ranging from 0 to 65535 (in hours) can be set.

NOTE

1 If a setting operation is attempted when the item name is not registered, the warning "EDIT REJECTED" is issued.
2 If a value exceeding the valid range is entered, the warning "DATA IS OUT OF RANGE" is issued.
3 If soft key [ERASE] or [TYPE] is pressed, the warning "EDIT REJECTED" is issued.

- Remaining time

The period of time left until the time for replacement is reached by count down operation is indicated.
If the percentage of the remaining time to the life time has reached the value (%) specified by parameter No. 8911, or smaller, the remaining time is displayed in red.
Even after the life time has expired, count down operation continues.

Move the cursor to the remaining time of a target registered number, type a remaining time, then press soft key [INPUT] (or the  key). The remaining time is then set.

When soft key [+INPUT] is pressed, the entered value can be added to the remaining time already set.
A value ranging from 0 to (life time) can be set.
When soft key [ERASE] then soft key [EXEC] are pressed, the same value as the life time is set.

NOTE
1 If a setting operation is attempted when the item name or life time is not registered, the warning “EDIT REJECTED” is issued.
2 When a value exceeding the valid range is entered, the warning “DATA IS OUT OF RANGE” is issued.
3 If horizontal soft key [TYPE] is pressed, the warning “EDIT REJECTED” is issued.

- Count type

As the count type, select the way of counting.
Place the cursor on the count type of a target registration number, then press soft key [TYPE]. Count types are displayed as soft keys as shown below. Select one of these soft keys, then press soft key [EXEC].

Soft key	Meaning	Indication
[NO CNT]	Counting is not performed (stopped)	———————
[ALL]	Counting is performed at all times	All times
[POWER ON]	Counting is performed when power is on.	When power is on
[RUN]	Counting is performed when operation is in progress.	When operation is in progress
[CUT]	Counting is performed when cutting is being performed.	When cutting is being performed

NOTE
1 If a setting operation is attempted when the item name or life time is not registered, the warning “EDIT REJECTED” is issued.
2 Soft keys [INPUT] and [+INPUT] have no effect.
3 When counting is performed at all times, a 24-hour error is generated in leap year.
4 If soft key [ERASE] is pressed, the warning “EDIT REJECTED” is issued.

- Path number

When [RUN] or [CUT] is set to the count type in multi path, path number can be set.
Make the cursor to the count type of a target registration number, then input path number and press soft key [INPUT].
The setting range is 0-(max path number).
When the specified path is operating or cutting, counting is performed.
If 0 is set, when either of path is operating or cutting, counting is perfomed.

NOTE
1 If [NO CNT], [ALL] or [POWER ON] is set to count type, "--" is displayed. In this case, if the setting is operated, the warning "EDIT REJECTED" is issued.
2 If the setting is beyond the valid data range, the warning "DATA IS OUT OF RANGE" is issued.
3 If soft key [TYPE] is pressed, the warning "EDIT REJECTED" is issued.
4 If soft key [ERASE] is pressed, the warning "EDIT REJECTED" is issued. The soft key [+INPUT] is invalid.

Registration from a program

The item name, life time, remaining time, count type and path number can be registered in the status screen and the setting screen by executing the program of the following format.

Format

G10 L60 Px [n] Aa Rr Qpt

X : Registration number

n : Item name, format: [alphanumeric-characters*2-byte-characters*alphanumeric-characters]

a : life time

r : remaining time

p : path number

t : count type

The meaning of the value of the count type is as follow.

0 : NO COUNT

1 : ALL TIMES

2 : POWER ON

3 : RUNNING

4 : CUTTING

The range of the path number is 0 – (max path number).

EX.)

Q24 : The path number is 2, and the count type is "CUTTING"

Q103 : The path number is 10, and the count type is "RUNNING"

Machine menu screen

On the machine menu screen, the names of consumables of the machine are registered. From this screen, item names can be added to the status screen. For the method of addition to the status screen, see the description of the status screen.

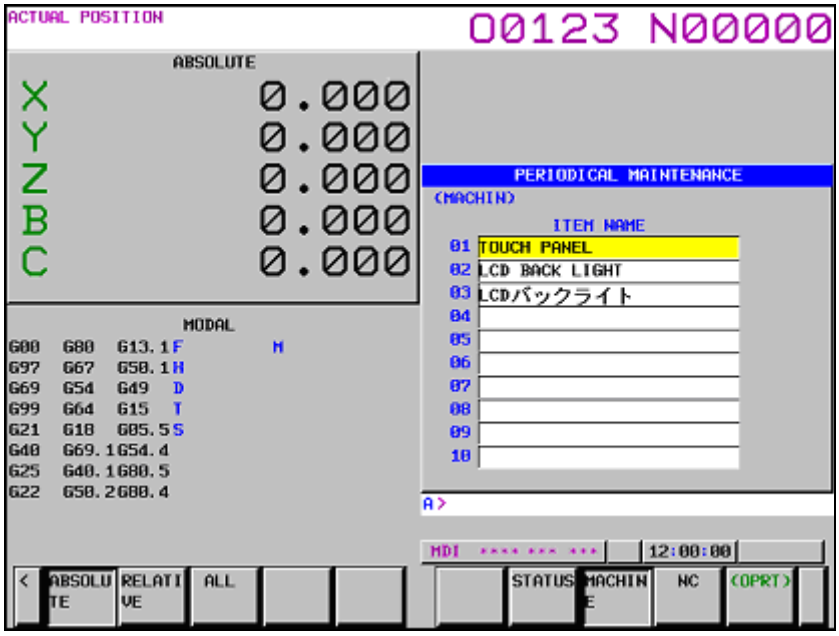


Fig. 12.4.11 (c) Machine menu screen

- Displaying the screen

1 When the status screen is displayed, press soft key [MACHINE].

On the machine menu screen, item names can be registered using one of the following two methods:

- Registration from a program
- Registration using the MDI keys

- Registration from a program

By executing a program having the following format, an item name can be registered in the menu of the machine:

Format

G10 L61 Px [n]

X Registration number

n Item name, format: [alphanumeric-characters*2-byte-characters*alphanumeric-characters]

- Registration using the MDI keys

An item name can be registered on the menu screen of the machine by typing the item name in the format shown below on the machine menu screen, then pressing soft key [INPUT] (or the INPUT key).

When soft key [+INPUT] is pressed, the typed characters can be added to an already registered item name.

Format

Alphanumeric-characters*2-byte-characters*alphanumeric-characters

Two-byte character codes must conform to FANUC codes. (See Appendix, "FANUC 2-BYTE CHARACTER CODE TABLE".)
When typing 2-byte characters, type an asterisk "*" before and after the character codes. An item name to be registered must be up to 24 characters long if it consists of alphanumeric characters only; or it must be up to 12 characters long if it consists of 2-byte characters only.

Example:

To register "LCD バックライト", enter the following:
>LCD*110E10F410CC114010B610FE*_

NOTE
1 An asterisk "*" is used as a control code, so it cannot be used in item names. In addition, characters "[", "]", "(", and ")" cannot be used in item names.
2 When an item name consisting of both alphanumeric characters and 2-byte characters is registered, the warning “DATA IS OUT OF RANGE” may be output.
3 When a blank item name is selected on the machine screen, the warning “EDIT REJECTED” is issued.

To delete a registered item name, move the cursor to the item name, press soft key [ERASE], then press soft key [EXEC].

NC menu screen

From this screen, an item name can be registered on the status screen. For the method of registration to the status screen, see the description of the status screen.

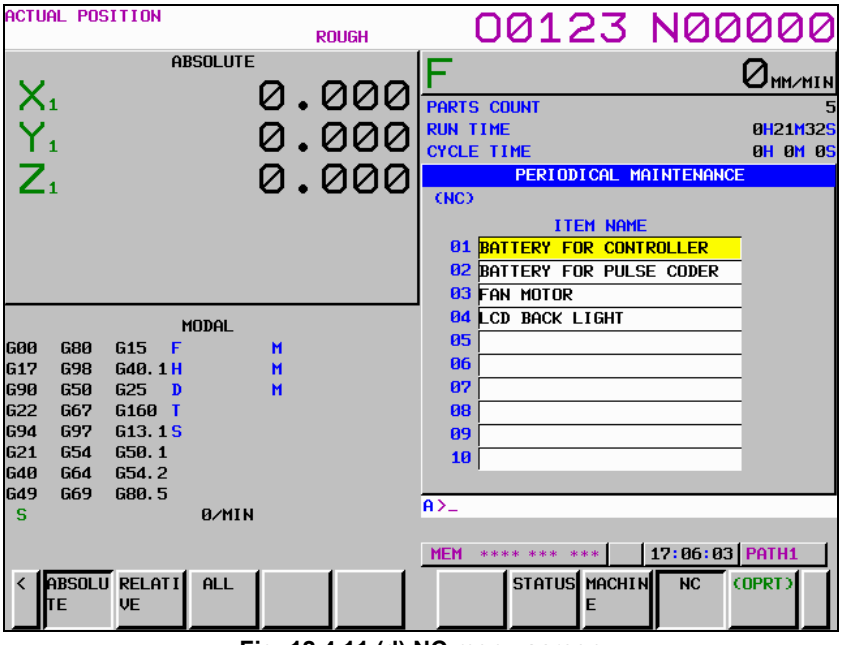


Fig. 12.4.11 (d) NC menu screen

- Displaying the screen

1 When the status screen is displayed, press soft key [NC].

NOTE
On the NC screen, the registration, deletion, and I/O of item names cannot be performed.
When a blank item name is selected, a blank is set.

12.4.12 System Configuration Screen

The system configuration screen provides information about the types of installed hardware and software.

Procedure for displaying the screen

Procedure

1 Press the function key 

to display a screen that shows parameters and other information.

2 Press soft key [SYSTEM]. The system configuration screen is displayed.

There are two types of system configuration screens: the hardware configuration screen and the software configuration screen. The screen display can switch between these screens by using 

and 

. When soft key [SERVO INFO] or [SPINDLE INFO] is pressed, information about the connected servo system or spindles is displayed.

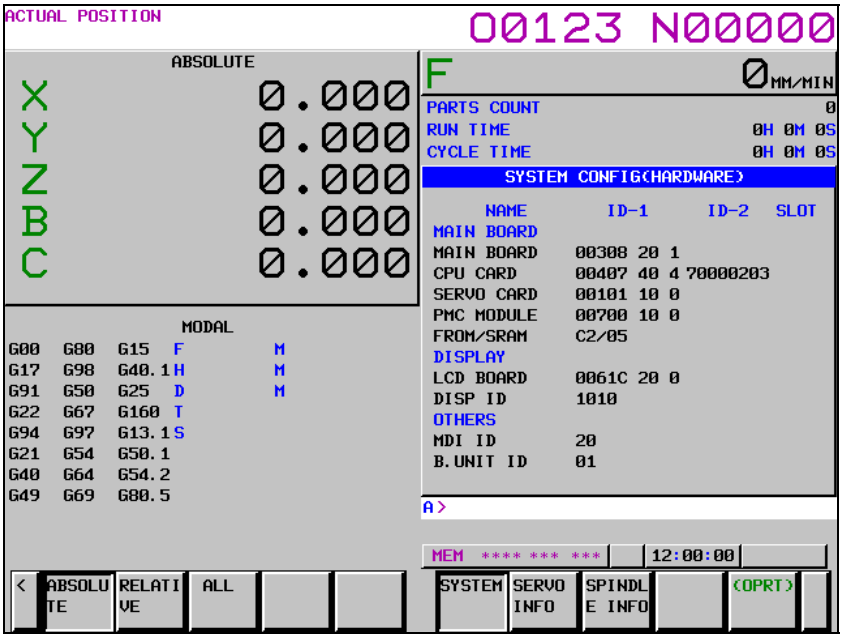


Fig. 12.4.12 (a) System configuration screen

Hardware configuration screen

This screen shows the names and IDs of the hardware used by the NC.

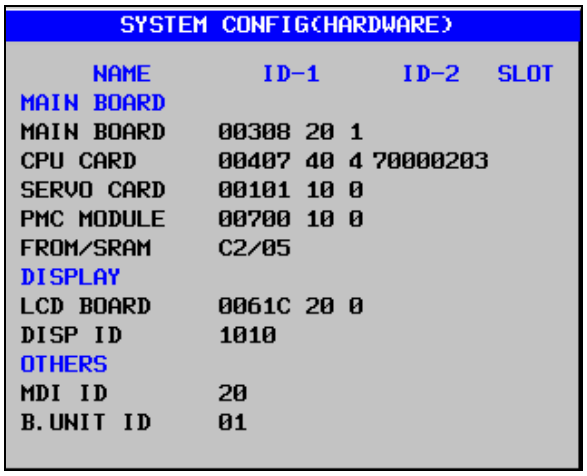


Fig. 12.4.12 (b) Hardware configuration screen

Software configuration screen

This screen shows the names and series/editions of the software used by the NC.

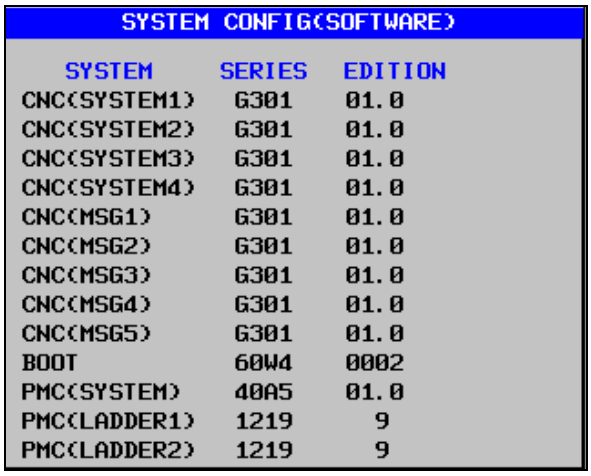


Fig. 12.4.12 (c) Software configuration screen

Servo information screen

When a servo system is connected to the NC, ID information of the connected servo devices (servo motors and servo amplifiers) can be displayed on the NC.

Displaying the screen

1 When the system configuration screen is displayed, press soft key [SERVO INFO].
2 The servo information screen is displayed.

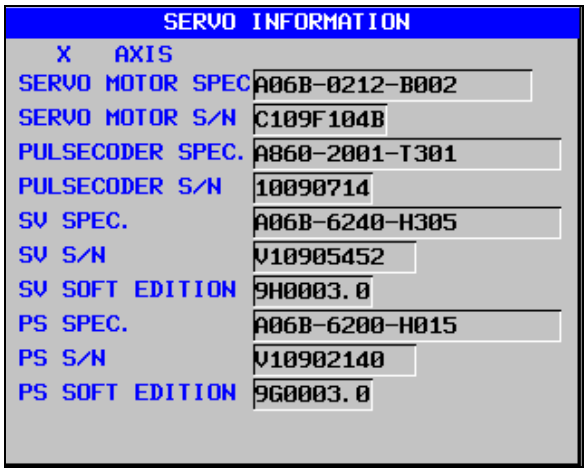


Fig. 12.4.12 (d) Servo information screen

Spindle information screen

When a spindle system is connected to the NC, the ID information of the connected spindle devices (spindle motors and spindle amplifiers) can be displayed on the NC.

Displaying the screen

1 When the system configuration screen is displayed, press soft key [SPINDLE INFO].
2 The spindle information screen is displayed.

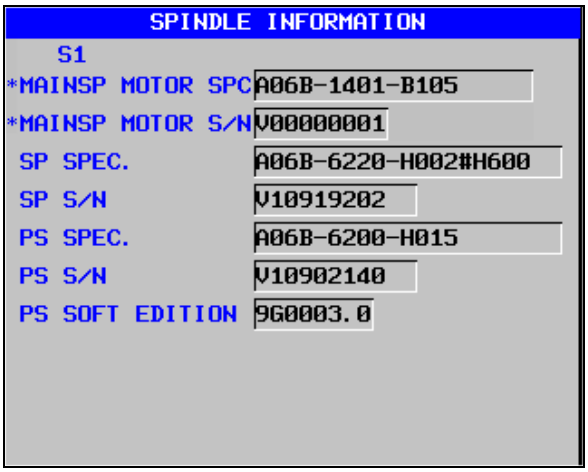


Fig. 12.4.12 (e) Spindle information screen

12.4.13 Power Consumption Monitoring Screen

The electric power data of servo axis and spindle axis consumption and regeneration can be displayed.

Display of power consumption monitoring screen

1 Press function key 

.

2 Press continuous menu key  several times until soft key [POWER MONIT] appears.
4 Press soft key [POWER MONIT].

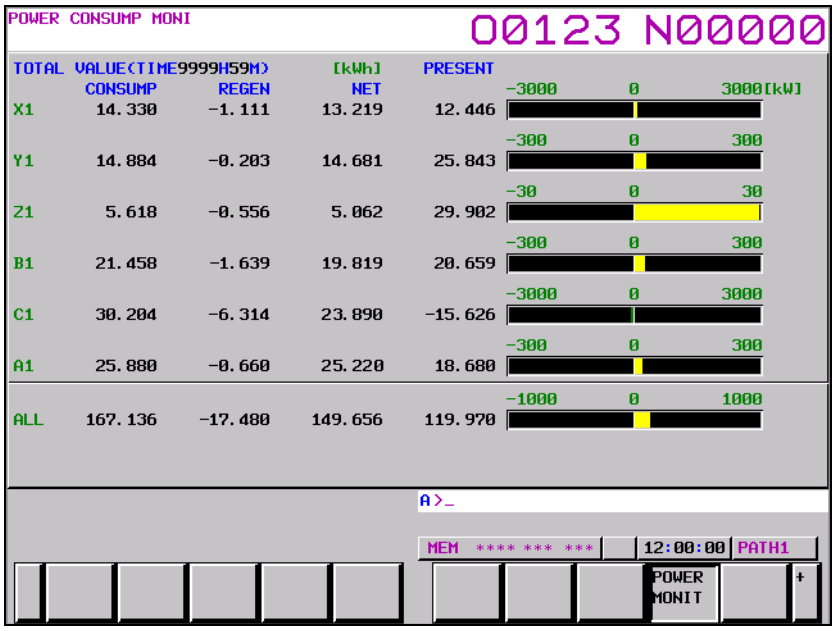


Fig. 12.4.13 (a) Power consumption monitoring screen for 10.4-inch display unit

Procedure for 8.4-inch display unit

1 Press function key 

.

2 Press continuous menu key  several times until soft key [POWMON] appears.

3 Press soft key [POWMON].

4 Press soft key [TOTAL] to display value of integral power consumption.

5 Press soft key [GRAPH] to display bar-graph.

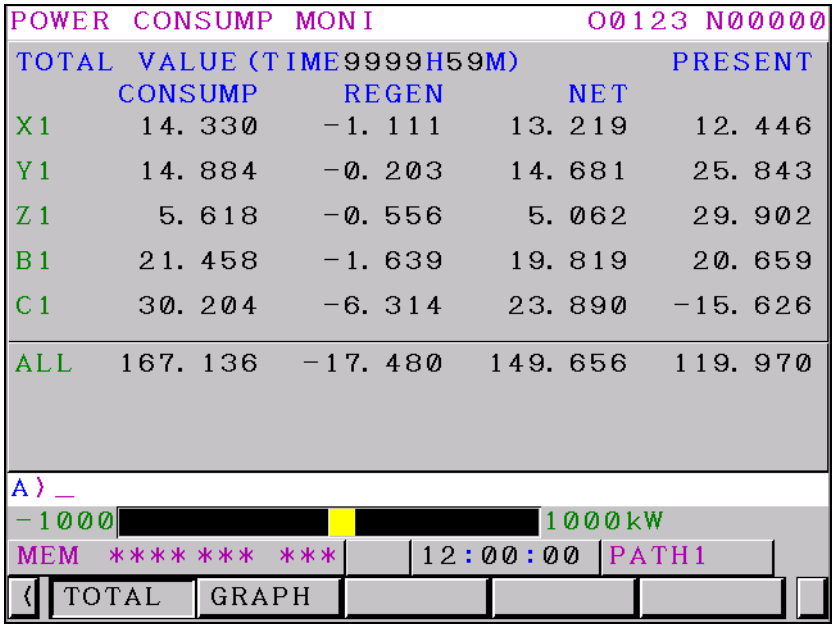


Fig. 12.4.13 (b) Power consumption monitoring screen (TOTAL) for 8.4-inch display unit

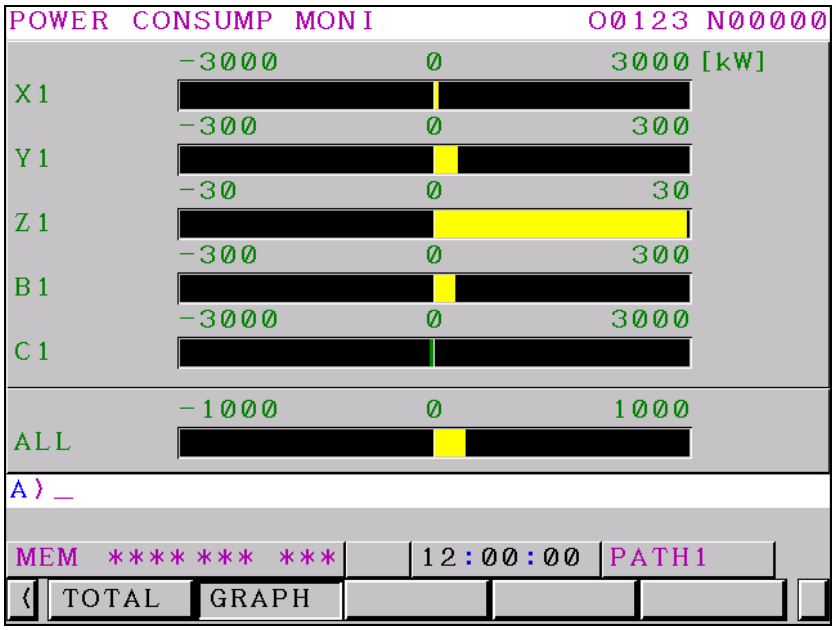


Fig. 12.4.13 (c) Power consumption monitoring screen (Bar-graph) for 8.4-inch display unit

Operation of power consumption monitoring screen

Switch of display axis

When information on all axes is not displayed, the page is switched with the MDI key <PageUp> or <PageDown>.

Explanation

TIME

Integrating time of power consumption is displayed.

Axis name

Axis name of servo and spindle is displayed. "ALL" means the total amount of power consumption.

CONSUMP

Integral power consumption is displayed.

REGEN

Integral power regeneration is displayed.

NET

Integral net amount of power consumption is displayed.
The net amount of power consumption = power consumption – power regeneration.

PRESENT

Present net power consumption is displayed. A negative value is displayed about the net power consumption while the power is regenerating like the axis is decelerating, etc.

Bar-graph

Current net power consumption is displayed as a bar-graph.
The scale of bar-graph can be specified in the parameter (No.11392 in case of the servo, No.11393 in case of the spindle).
The scale of the entire power consumption bar-graph can be specified in the parameter (No.11371).
The bar expands from the center to right while consuming the electric power. The bar expands from the center to left while regenerating the electric power.

Screens of a 15/19-inch display unit

12.4.14 Displaying and Setting Parameters (15/19-inch Display Unit)

When the CNC and machine are connected, parameters are set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor. The setting of parameters depends on the machine. Refer to the parameter list prepared by the machine tool builder. Normally, the user need not change parameter setting.

Procedure for displaying and setting parameters

Procedure

1 Set 1 for PARAMETER WRITE to enable writing. See the procedure for enabling/disabling parameter writing described below.

2 Press function key 

.

3 Press vertical soft key [PARAM] to display the parameter screen.

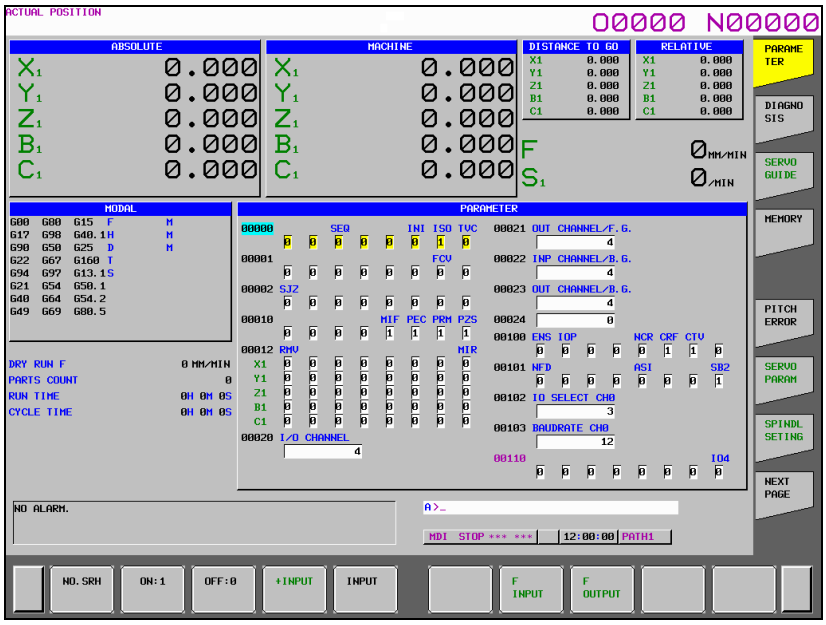


Fig. 12.4.14 (a) PARAMETER screen (15-inch display unit)

4 Move the cursor to the parameter number to be set or displayed in either of the following ways:

- Enter the parameter number and press horizontal soft key [NO.SRH] .
- Move the cursor to the parameter number using the page keys, 

and 

, and cursor keys, 
, 
, 
, and 
.

5 To set the parameter, enter a new value with numeric keys and press horizontal soft key [INPUT]. The parameter is set to the entered value and the value is displayed.

6 Set 0 for PARAMETER WRITE to disable writing.

Procedure for enabling/displaying parameter writing

Procedure

1 Select the MDI mode or enter state emergency stop.

2 Press function key 

.

3 Press vertical soft key [SETTING] to display the setting screen.

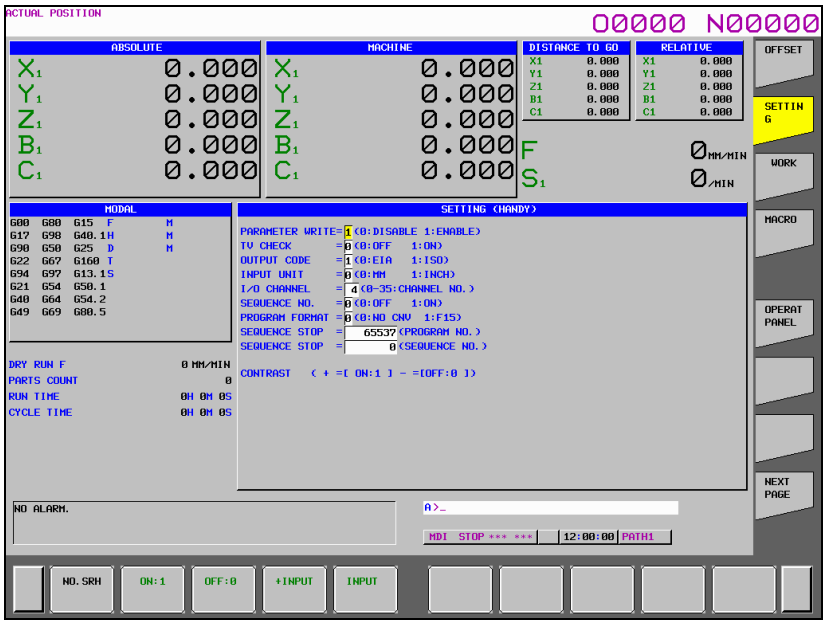


Fig. 12.4.14 (b) SETTING screen (15-inch display unit)

4 Move the cursor to PARAMETER WRITE using cursor keys.
5 Press horizontal soft key [ON:1] to enable parameter writing.
At this time, the CNC enters the alarm state SW0100.
6 After completing parameter setting, move the cursor to PARAMETER WRITE on the setting screen again, and press horizontal soft key [OFF:0].
7 Depress the RESET key to release the alarm condition.

When an alarm PW0000 occurred, however, it is not released unless the power is turned off and back on.

Explanation

- Setting parameters with external input/output devices

See Chapter, “DATA INPUT/OUTPUT” for setting parameters with external input/output devices such as the memory card.

- Parameters that require turning off the power

Some parameters are not effective until the power is turned off and on again after they are set. Setting such parameters causes alarm PW0000. In this case, turn off the power, then turn it on again.

- Parameter list

Refer to the Parameter Manual (B-64490EN) for the parameter list.

- Setting data

Some parameters can be set on the setting screen if the parameter list indicates "Setting entry is acceptable". Setting 1 for PARAMETER WRITE is not necessary when three parameters are set on the setting screen.

12.4.15 Servo Parameters (15/19-inch Display Unit)

This subsection describes the initialization of digital servo parameters performed, for example, at the time of field tuning of the machine tool.

Procedure for servo parameter setting

Procedure

1 Turn on the power in the emergency stop state.
2 Set bit 0 (SVS) of parameter No. 3111 to 1 to display servo setting and tuning screens.
3 Press function key SYSTEM then vertical soft key [SERVO PARAM]. The following screen appears:

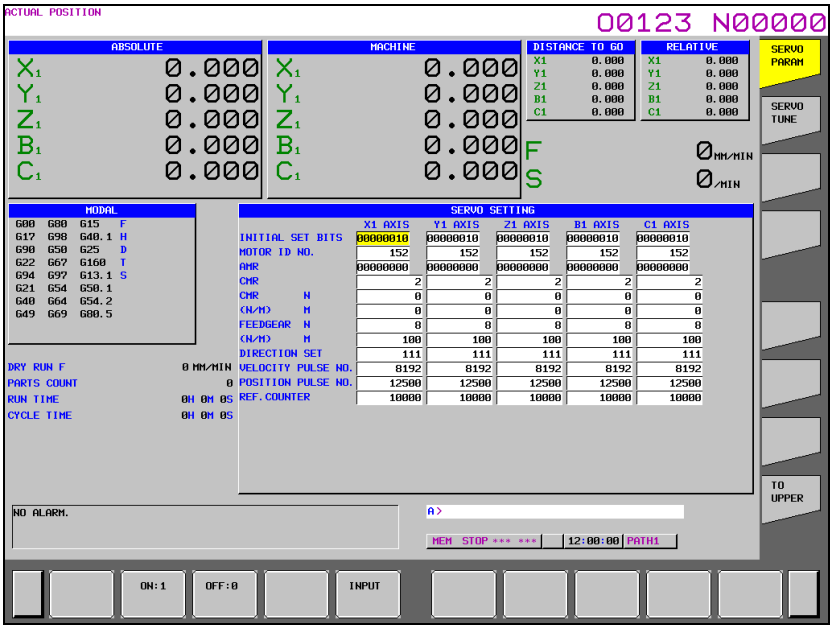


Fig. 12.4.15 (a) Servo parameter setting screen (15-inch display unit)

4 With the page keys and cursor keys, move the cursor to the position of data to be set or modified.
5 Key in a desired value then press horizontal soft key [INPUT].

12.4.16 Servo Tuning (15/19-inch Display Unit)

Data related to servo tuning is displayed and set.

Procedure for servo tuning

Procedure

1 Turn on the power in the emergency stop state.
2 Set bit 0 (SVS) of parameter No. 3111 to 1 to display servo setting and tuning screens.
3 Press function key 

then vertical soft key [SERVO PARAM].
4 Press vertical soft key [SERVO TUNE] to select the servo tuning screen. The servo tuning screen appears:

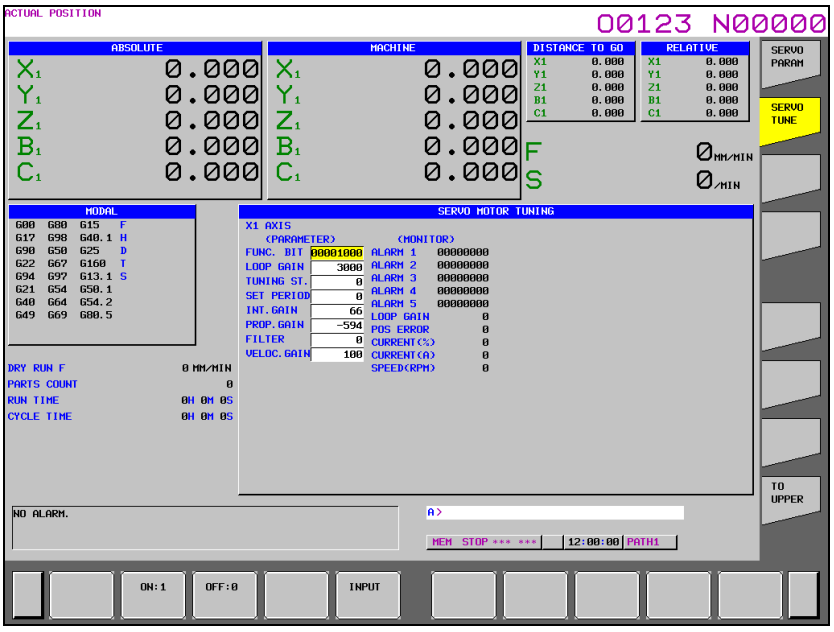


Fig. 12.4.16 (a) Servo tuning screen (15-inch display unit)

5 With the page keys and cursor keys, move the cursor to the position of data to be set or modified.
6 Key in a desired value then press horizontal soft key [INPUT].

12.4.17 Spindle Setting (15/19-inch Display Unit)

Parameters related to spindles are set and displayed. In addition to the parameters, related data can be displayed. Screens for spindle setting, spindle tuning, and spindle monitoring are provided.

Setting spindle parameters

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display spindle setting and tuning screens.

2 Press function key 

then vertical soft key [SPINDL SETING].

3 Press vertical soft key [SPINDL SETING] to select the spindle setting screen.
The spindle setting screen appears:

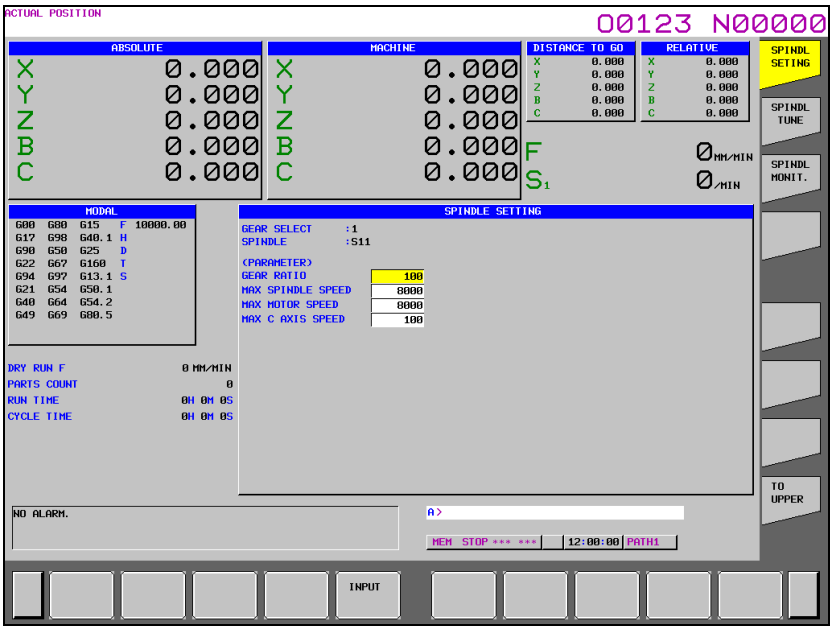
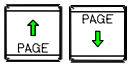
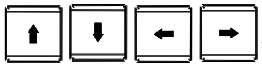


Fig. 12.4.17 (a) Spindle setting screen (15-inch display unit)

4 With the page keys 

and cursor keys , move the cursor to the position of data to be set or modified.

5 Key in a desired value then press horizontal soft key [INPUT].

12.4.18 Spindle Tuning (15/19-inch Display Unit)

Spindle tuning data is displayed and set.

Setting for spindle tuning

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display spindle setting and tuning screens.

2 Press function key 

then vertical soft key [SPINDL SETING].

3 Press vertical soft key [SPINDL TUNE] to select the spindle tuning screen.
The spindle tuning screen appears:

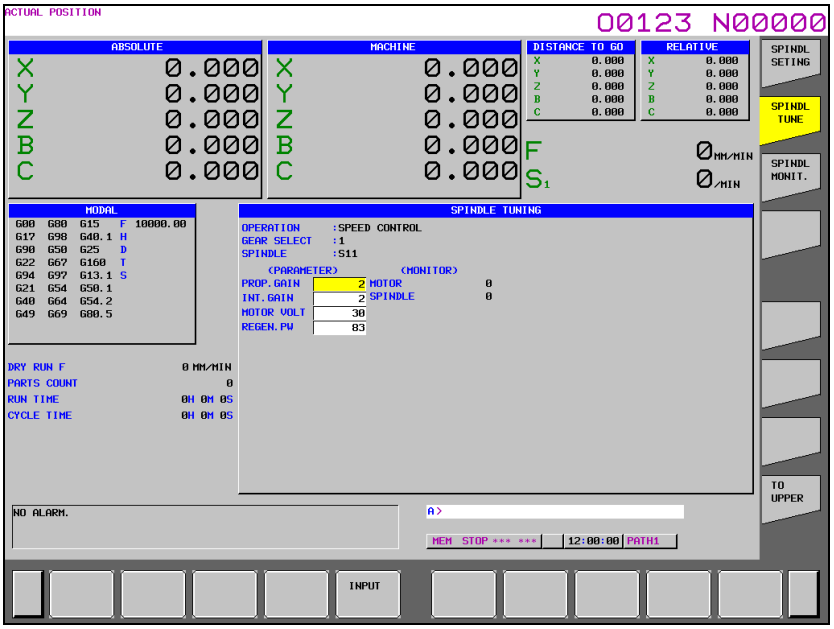
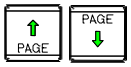
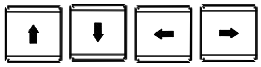


Fig. 12.4.18 (a) Spindle tuning screen (15-inch display unit)

4 With the page keys 

and cursor keys , move the cursor to the position of data to be set or modified.

5 Key in a desired value then press horizontal soft key [INPUT].

12.4.19 Spindle Monitor (15/19-inch Display Unit)

Spindle-related data is displayed.

Displaying the spindle monitor

Procedure

1 Set bit 1 (SPS) of parameter No. 3111 to 1 to display spindle setting and tuning screens.

2 Press function key 

then vertical soft key [SPINDL SETING].

3 Press vertical soft key [SPINDL MONIT.] to select the spindle monitor screen.
The spindle monitor screen appears:

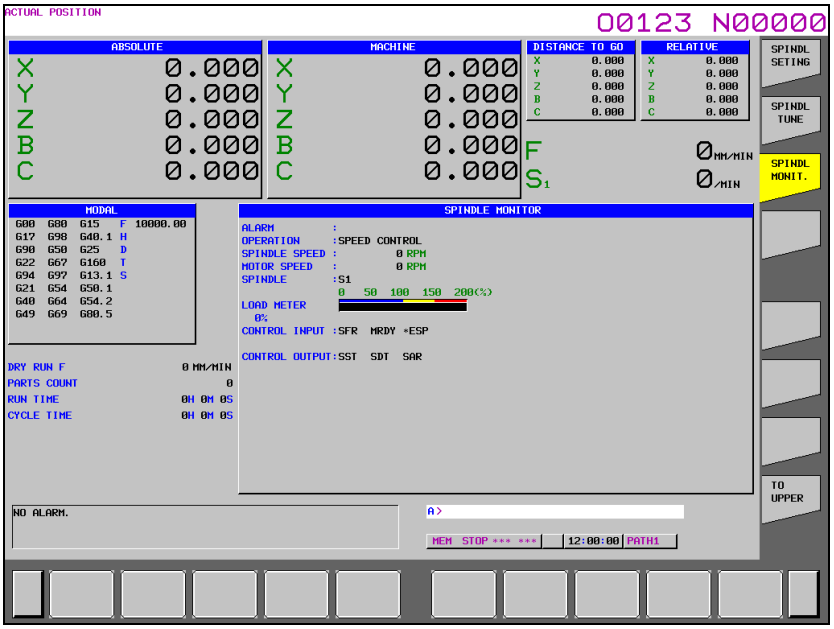


Fig. 12.4.19 (a) Spindle monitor screen (15-inch display unit)

12.4.20 Color Setting Screen (15/19-inch Display Unit)

Screen colors can be set on the color setting screen.

Displaying the color setting screen

Procedure

1 Press function key SYSTEM.
2 Press vertical soft key [NEXT PAGE] several times to display vertical soft key [COLOR].
3 Press vertical soft key [COLOR] to display the color setting screen.

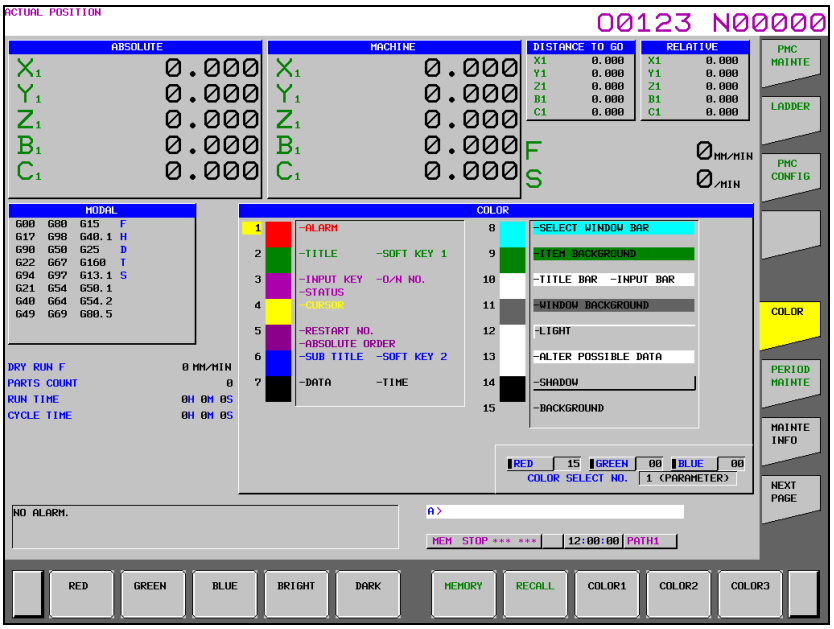


Fig. 12.4.20 (a) Color setting screen (15-inch display unit)

Procedure for operating the color setting screen

- Modifying the color (color palette values)

1 The following horizontal soft keys are displayed:



2 Move the cursor to a color number whose color palette values are to be modified.
The current color palette value set for each primary color is displayed.
3 Select a primary color whose setting is to be modified, with the corresponding horizontal soft key [RED], [GREEN], or [BLUE].
More than one primary color can be selected at the same time.
Each time horizontal soft key [RED], [GREEN], or [BLUE] is pressed, the operation soft key toggles between selection and deselection.
4 Select horizontal soft key [BRIGHT] or [DARK] to modify the brightness of the selected prime color(s).

- Storing the color (color palette values)

The set color palette values can be stored.

1 Press horizontal soft key [COLOR1], [COLOR2], or [COLOR3] to select a storage area.



COLOR1 Standard color data parameters (Nos. 6581 to 6595)
COLOR2 Parameters (Nos. 10421 to 10435)
COLOR3 Parameters (Nos. 10461 to 10475)

2 Press horizontal soft key [MEMORY]. The horizontal soft key display switches to the following:



3 Pressing horizontal soft key [EXEC] stores the current color palette settings in the selected area.
Pressing horizontal soft key [CAN] or the leftmost key does not store the current color palette settings in the selected area.

- Calling the color (color palette values)

1 Press horizontal soft key [COLOR1], [COLOR2], or [COLOR3] to select a storage area where color palette values are stored.



2 Press horizontal soft key [RECALL]. The horizontal soft key display switches to the following:



3 Pressing horizontal soft key [EXEC] calls the color palette values from the selected area to enable the color to be modified. This operation is invalid if no color palette values are stored.
Pressing horizontal soft key [CAN] or the leftmost key does not call the color palette values from the selected area.

NOTE
1 Immediately after the power is turned on, the settings of COLOR1 (parameters) are used for display. If nothing is stored for COLOR1, the default colors are used for display.
2 Do not modify the color setting data parameters directly by MDI key input. When modifying the standard color data, be sure to perform a storage operation on the color setting screen.

12.4.21 Machining Parameter Tuning (15/19-inch Display Unit)

12.4.21.1 Machining parameter tuning (AI contour)

In AI contour control, by setting a velocity-emphasized parameter set and precision-emphasized parameter set and setting the precision level matching a machining condition such as rough machining or finish machining on the precision level setting screen or by programming, the parameters suitable for the condition can be automatically calculated to perform machining.
On this screen, the parameter sets for emphasis on velocity (precision level 1) and emphasis on precision (precision level 10) can be set.
Set the following parameters:

- Acceleration rate of acceleration/deceleration before interpolation
- Acceleration change time (bell-shaped)
- Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control
- Allowable acceleration change value for each axis in acceleration change under jerk control in successive linear interpolation operations

- Ratio of the change time of the rate of change of acceleration in smooth bell-shaped acceleration/deceleration before interpolation
- Allowable acceleration rate
- Acceleration rate of acceleration/deceleration after interpolation
- Corner speed difference
- Maximum feedrate
- Items that can be set freely (2 items)

For details of each parameter, see the descriptions of AI contour control and jerk control.
By setting bit 0 (MPR) of parameter No. 13601 to 1, this screen can be hidden.
For the method of setting a precision level, see the description of the precision level selection screen in Subsection, “Precision Level Selection”.

Procedure for machining parameter tuning

Procedure

1 Set the MDI mode.

2 Press function key SYSTEM.

3 Press vertical soft key [NEXT PAGE] several times to display vertical soft key [MCHN TUNING]. The machining parameter tuning screen appears:

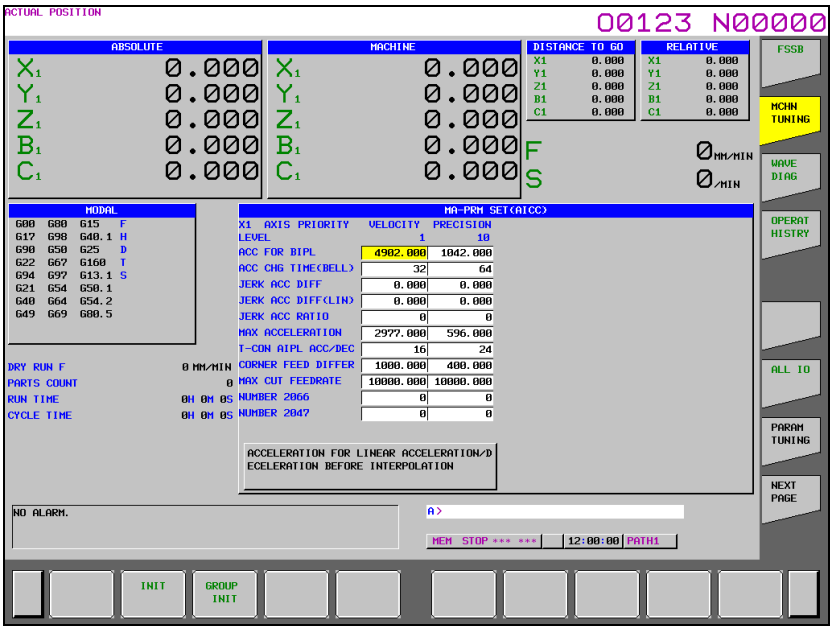


Fig. 12.4.21.1 (a) Machining parameter tuning screen (AI contour) (15-inch display unit)

4 Move the cursor to the position of a parameter to be set, as follows:
Press page key PAGE or PAGE, and cursor keys, and /or to move the cursor to the parameter.

5 Key in desired data then press the INPUT key on the MDI unit.

6 When data is input, a RMS value is found according to the precision level parameters. (The precision level can be changed on the precision level selection screen or parameter setting screen.) If a RMS value calculation fails, a warning (indicating that automatic setting failed) is displayed.

7 Repeat steps 4 and 5 until all machining parameters are set.

8 In addition to the setting method described above, a parameter setting method using horizontal soft keys is available. Pressing horizontal soft key [INIT] displays the standard value (recommended by FANUC) of the item selected by the cursor in the key input buffer. Pressing horizontal soft key [EXEC] initializes the item to the standard value. Pressing horizontal soft key [GROUP INIT] initializes all items of a group (emphasis on velocity or emphasis on precision) selected by the cursor to the standard values.

The table below indicates the initial settings.

Table 12.4.21.1 (a) Initial settings

Setting item	AI contour control		Unit
	Emphasis on velocity (LV1)	Emphasis on precision (LV10)	
Acceleration rate of acceleration/deceleration before interpolation <ACC FOR BIPL>	4902.000	1042.000	mm/sec^2
Acceleration change time (bell-shaped) <ACC CHG TIME(BELL)>	32	64	msec
Allowable acceleration change value <JERK ACC DIFF>	0	0	mm/sec^2
Allowable acceleration change value in successive linear interpolation operations <JERK ACC DIFF(LIN)>	0	0	mm/sec^2
Ratio of the change time of the jerk control <JERK ACC RATIO>	0	0	%
Allowable acceleration rate <MAX AACELERATION>	2977.000	596.000	mm/sec^2
Time constant for acceleration/deceleration after interpolation <T-CON AIPL ACC/DEC>	24	24	msec
Corner speed difference <CORNER FEED DIFFER>	1000	400	mm/min
Maximum cutting speed <MAX CUT FEEDRATE>	10000	10000	mm/min

Explanation

- Look-ahead acceleration/deceleration before interpolation

Set an acceleration rate for a linear portion in look-ahead acceleration/deceleration before interpolation.
Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13610 (velocity-emphasized parameter)
Parameter No. 13611 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1660: Maximum allowable acceleration rate for each axis in acceleration/deceleration before interpolation

- Acceleration change time (bell-shaped)

Set a time constant for a bell-shaped portion in acceleration/ deceleration before look-ahead interpolation.
Unit of data: ms

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13612 (velocity-emphasized parameter)
Parameter No. 13613 (precision-emphasized parameter)

When bit 7 of parameter No. 13600 is set to 1, the parameter set on the machining parameter tuning screen is reflected in the following parameters for which the valid data range is extended to the range from 0 to 200:
Parameter No. 13662 (velocity-emphasized parameter)
Parameter No. 13663 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1772: Time constant for bell-shaped look-ahead acceleration/deceleration before interpolation of constant acceleration time type

⚠ CAUTION
A set time constant is applied to all axes. So, a modification made to this item changes the settings for all axes.

- Allowable acceleration change value in velocity control based on acceleration change under jerk control

Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

Set an allowable acceleration change value per ms for each axis in velocity control based on acceleration change under jerk control.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13614 (velocity-emphasized parameter)
Parameter No. 13615 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1788: Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control

NOTE
This setting item is displayed only when the jerk control function is enabled.

- Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations

Unit of data: mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

Set an allowable acceleration change value per ms for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13616 (velocity-emphasized parameter)
Parameter No. 13617 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1789: Allowable acceleration change value for each axis in velocity control based on acceleration change under jerk control in successive linear interpolation operations

⚠ CAUTION
1 For an axis with 0 set in this parameter, the parameters (allowable acceleration change value in velocity control based on acceleration change under jerk control: No. 13614, No.13615) are valid.
2 For an axis with 0 set in the parameter (allowable acceleration change value in velocity control based on acceleration change under jerk control: No. 13614, No.13615), velocity control based on acceleration change is disabled, so that this parameter has no effect.

NOTE
This setting item is displayed only when the jerk control function is enabled.

- Ratio of the change time of the jerk control in smooth bell-shaped acceleration/deceleration before interpolation

Unit of data: %

Set the ratio (in %) of the change time of jerk control to the change time of acceleration in smooth bell-shaped acceleration/deceleration before interpolation.

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13618 (velocity-emphasized parameter)
Parameter No. 13619 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1790: Ratio of the change time of the jerk control in smooth bell-shaped acceleration/deceleration before interpolation

NOTE
This setting item is displayed only when the jerk control function is enabled.

- Allowable acceleration rate

Set an allowable acceleration rate in acceleration-based speed determination.
Unit of data: mm/sec^2, $inch/sec^2$, deg/sec^2 (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13620 (velocity-emphasized parameter)
Parameter No. 13621 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1735: Allowable acceleration rate for each axis applicable to the deceleration function based on acceleration in circular interpolation
Parameter No. 1737: Allowable acceleration rate for each axis applicable to the deceleration function based on acceleration in AI contour control

⚠ CAUTION
When bit 0 (MCR) of parameter No. 13600 is set to 1, the deceleration function based on acceleration in circular interpolation is not set.

- Time constant for acceleration/deceleration after interpolation

Set a time constant for acceleration/deceleration after interpolation.
Unit of data: ms

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13622 (velocity-emphasized parameter)
Parameter No. 13623 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1769: Time constant for acceleration/deceleration after cutting feed interpolation

- Corner speed difference

Set an allowable corner speed difference used for speed determination.
Unit of data: mm/sec, inch/sec, deg/sec (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13624 (velocity-emphasized parameter)
Parameter No. 13625 (precision-emphasized parameter)

Moreover, the following parameter is also set according to the precision level:
Parameter No. 1783: Allowable speed difference for each axis in automatic corner deceleration based on speed difference

- Maximum cutting speed

Set a maximum cutting speed for each axis.
Unit of data: mm/sec, inch/sec, deg/sec (machine unit)

The parameter set on the machining parameter tuning screen is reflected in the following parameters:
Parameter No. 13626 (velocity-emphasized parameter)
Parameter No. 13627 (precision-emphasized parameter)

Moreover, the following parameter is also set from the precision level:
Parameter No. 1432: Maximum cutting feedrate for each axis in the AI contour control mode

- Arbitrary items

Two arbitrary parameters can be registered. Each item can correspond to a CNC parameter or servo parameter. A parameter number corresponding to each item is to be specified with parameters.

As indicated below, set the parameters for corresponding parameter numbers, velocity-emphasized parameters (precision level 1), and precision-emphasized parameters (precision level 10).

Table 12.4.21.1 (b) Parameters related to arbitrary items

	Corresponding parameter number	Setting of velocity-emphasized (precision level 1) value	Setting of precision-emphasized (precision level 10) value
Arbitrary item 1	No.13628	No.13630	No.13632
Arbitrary item 2	No.13629	No.13631	No.13633

- Display
  Tuning target parameter numbers are displayed.

⚠ CAUTION
As arbitrary items, the numbers of the following parameters cannot be specified:
- Bit parameter
- Spindle parameters (Parameters Nos. 4000 to 4799)
- Real-type parameter
- Power-off parameter
- Nonexistent parameter

12.4.21.2 Machining parameter tuning (nano smoothing)

In nano smoothing, by setting a parameter set and setting the smoothing level matching a machining condition on the smoothing level selection screen or by programming, the parameters suitable for the condition can be automatically calculated to perform machining.
On this screen, the parameter sets for emphasis on precision (smoothing level 1) and emphasis on surface smoothing (smoothing level 10) can be set.
Set the following parameters:

- Tolerance

- Tolerance (rotary axis)

For details of each parameter, see the descriptions of nano smoothing.
By setting bit 0 (MPR) of parameter No. 13601 to 1, this screen can be hidden.
For the method of setting a smoothing level, see the description of the smoothing level selection screen in Subsection, “Smoothing level selection”.

NOTE
These setting items are displayed only when machining quality level adjustment is enabled.

Procedure for machining parameter tuning

1 Set the MDI mode.
2 Press function key SYSTEM.
3 Press vertical soft key [MCHN TUNING].
4 Press vertical soft key [NANO SMOOTH] to display the machining parameter tuning screen.

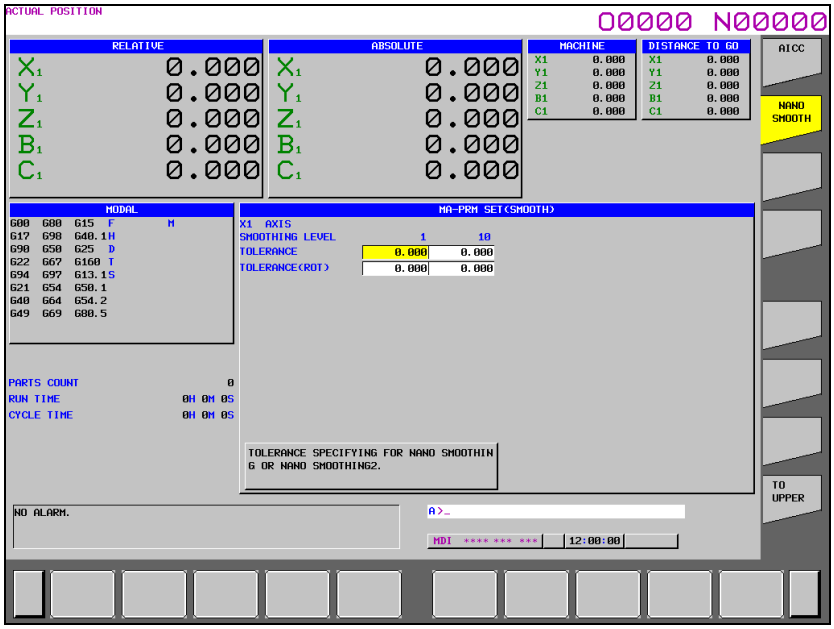


Fig. 12.4.21.2 (a) Machining parameter tuning screen (nano smoothing) (15-inch display unit)

5 Move the cursor to the position of a parameter to be set, as follows:
Press page key PAGE or PAGE, and cursor keys, and /or to move the cursor to the parameter.
6 Key in desired data then press the INPUT key on the MDI unit.
7 When data is input, a RMS value is found according to the smoothing level parameters. (The smoothing level can be changed on the smoothing level selection screen or parameter setting screen.) If a RMS value calculation fails, a warning (indicating that automatic setting failed) is displayed.
8 Repeat steps 5 and 6 until all machining parameters are set.

Explanation

- Tolerance

Set the value specified for the tolerance for nano smoothing.
Unit of data: mm, inch, degree (input unit)

The parameter set on the machining parameter tuning screen (smoothing) is reflected in the following parameters:
Parameter No. 11682 (smoothing level 1)
Parameter No. 11683 (smoothing level 10)

Moreover, the following parameter is also set according to the smoothing level:
Parameter No. 19541: Tolerance specified for nano smoothing

> ⚠ **CAUTION**
> Since the tolerance specified for nano smoothing is common to all axes, changing this item changes the setting for all axes.

- Tolerance (rotary axis)

Set the value specified for the tolerance for the rotary axes in nano smoothing 2.
Unit of data: mm, inch, degree (input unit)

The parameter set on the machining parameter tuning screen (smoothing) is reflected in the following parameters:
Parameter No. 11684 (smoothing level 1)
Parameter No. 11685 (smoothing level 10)

Moreover, the following parameter is also set according to the smoothing level:
Parameter No. 19547: Tolerance specified for rotary axes in nano smoothing 2

> ⚠ **CAUTION**
> This setting item is displayed only when the nano smoothing 2 option is enabled.

12.4.22 Displaying Memory Data (15/19-inch Display Unit)

The contents of the CNC memory can be displayed starting at a specified address.

Displaying memory data

Procedure

1 Set bit 0 (MEM) of parameter No. 8950 to 1 to display the memory contents display screen.
2 Press function key .
3 Press vertical soft key [MEMORY].
The following screen appears:

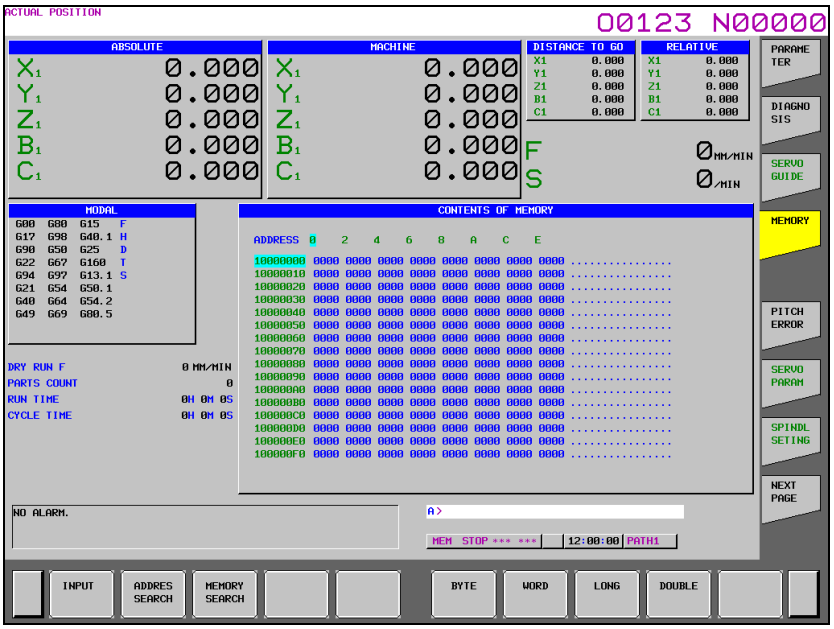


Fig. 12.4.22 (a) Memory contents display screen (15-inch display unit)

4 Key in a desired address (hexadecimal) then press horizontal soft key [ADDRES SEARCH]. Starting at the specified address, 256-byte data is displayed.
(Example: When you input 100000 then press [ADDRES SEARCH], data is displayed starting at 100000H.)

5 Display data can be switched by using page keys [PAGE↑] and [PAGE↓] and cursor keys [↑], [↓], [←], and/or [→].

6 By pressing horizontal soft key [BYTE], [WORD], [LONG], or [DOUBLE], select a data type to be displayed.

Explanation

A memory data display format can be selected from the following four options:

Byte display (1 byte in hexadecimal)
Word display (2 bytes in hexadecimal)
Long display (4 bytes in hexadecimal)
Double display (8 bytes in decimal: Double precision floating-point display)

One screen displays 256-byte memory data.

NOTE
1 When an address is input, "H" for representing "hexadecimal" need not be specified at the end of the address. If H is added, a warning is displayed to indicate that the format is invalid.
2 When word display is selected as the display format, an input address is rounded to a multiple of 2 bytes. When long display or double display is selected, an input address is rounded to a multiple of 4 bytes.

⚠ WARNING

1 If a memory address that must not be accessed in address search is input, a system alarm is issued.
When making an address search, check that the address is accessible and that the address is input correctly.
2 This function is designed for maintenance, and must not be used by general users.

12.4.23 Parameter Tuning Screen (15/19-inch Display Unit)

The parameter tuning screen is a screen for parameter setting and tuning designed to achieve the following:

1 The minimum required parameters that must be set when the machine is started up are collectively displayed to facilitate start-up of the machine.
2 The servo tuning screen, spindle tuning screen, and machining parameter tuning screen are displayed for smooth tuning.

The parameter tuning screen consists of a menu screen and several setting screens.

12.4.23.1 Displaying the menu screen and selecting a menu item (15/19-inch display unit)

The parameter tuning menu screen displays the following items:

[START UP]
- SYSTEM SETTING
- AXIS SETTING
- FSSB (SV AMP)
- FSSB (SP AMP)
- FSSB (AXIS)
- SERVO SETTING
- SPINDLE SETTING
- MISCELLANY

[TUNING]
- SERVO TUNING
- SPINDLE TUNING
- AICC TUNING

On the parameter tuning menu screen, one of the displayed items can be selected to display the corresponding screen. From each setting screen, you can return to the menu screen by performing a horizontal soft key operation.

NOTE
1 Some items may not be displayed, depending on the system configuration.
2 When bit 0 (SVS) of parameter No. 3111 is set to 0, "SERVO SETTING" and "SERVO TUNING" are not displayed. When bit 1 (SPS) of parameter No. 3111 is set to 0, "SPINDLE TUNING" is not displayed.

Displaying the menu screen and selecting a setting screen

Procedure

1 Set the MDI mode.
2 Switch the setting of "PARAMETER WRITE" to "ENABLED". For details, see the procedure for "PARAMETER WRITE" in Subsection, “Displaying and Setting Parameters (15/19-inch Display Unit)”.
3 Press function key 

.
4 Press vertical soft key [NEXT PAGE] several times then press vertical soft key [PARAM TUNING]. The following screen appears:

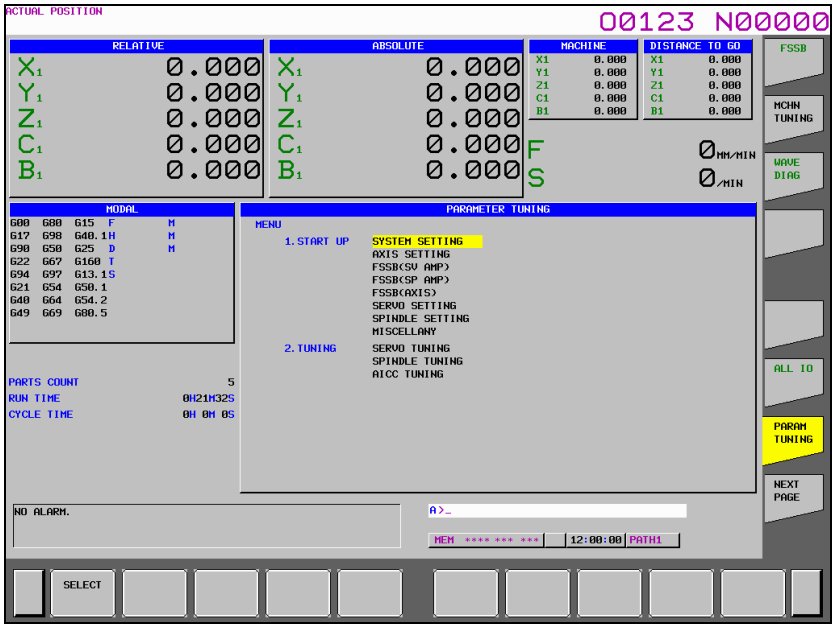


Fig. 12.4.23.1 (a) Parameter tuning menu screen (15-inch display unit)

5 Move the cursor to a desired item by pressing cursor key  or .

6 Press horizontal soft key [SELECT]. The screen display switches to the selected screen.

Returning to the menu screen

Procedure

1 Press horizontal soft key [SELECT] on the parameter tuning menu screen described in Subsection III-12.4.27.1. The screen and soft keys shown below (Fig. 12.4.23.1 (b)) are displayed. (The screen below (Fig. 12.4.23.1 (b)) is displayed when "AXIS SETTING" is selected.)

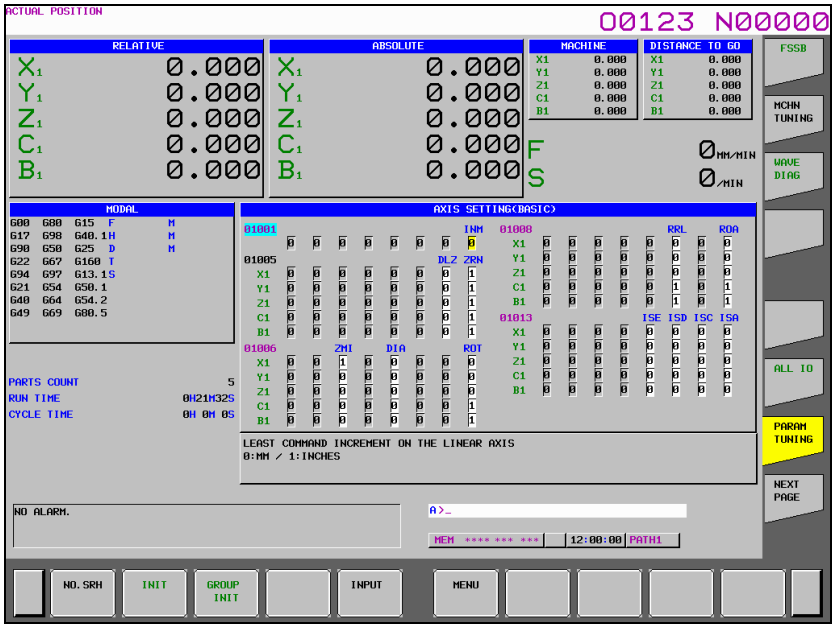


Fig. 12.4.23.1 (b) Axis setting screen (15-inch display unit)

2 Press horizontal soft key [MENU].
The screen display returns to the parameter tuning menu screen.

3 Upon completion of parameter setting, switch the setting of "PARAMETER WRITE" to "DISABLED".

NOTE
Part of the setting screens can be displayed also by using a vertical soft key for chapter selection. When these screens are displayed using the vertical soft key for chapter selection, it becomes impossible to return to the parameter tuning menu screen.

Explanation

- Items displayed with [START UP]

The items of [START UP] indicate the screens for setting the minimum required parameters for starting up the machine.

Table 12.4.23.1 (a) Items displayed with [START UP]

Display item	Description
SYSTEM SETTING	Screen for setting CNC parameters related to the entire system configuration
AXIS SETTING	Screen for setting CNC parameters related to axes, coordinates, feedrate, and acceleration/deceleration
FSSB (SV AMP)	FSSB servo amplifier setting screen
FSSB (SP AMP)	FSSB spindle amplifier setting screen
FSSB (AXIS)	FSSB axis setting screen
SERVO SETTING	Servo setting screen
SPINDLE SETTING	Screen for setting spindle-related parameters
MISCELLANY	Screen for setting parameters related to the allowable number of M code digits and whether to display the servo setting and spindle tuning screens

NOTE
Some items may not be displayed, depending on the system configuration.

- Items displayed with [TUNING]

The items of [TUNING] indicate the screens for servo, spindle, and high-speed high-precision machining tuning.

Table 12.4.23.1 (b) Items displayed with [TUNING]

Display item	Description
SERVO TUNING	Servo tuning screen
SPINDLE TUNING	Spindle tuning screen
AICC TUNING	Machining parameter tuning screen

NOTE
1 Some items may not be displayed, depending on the system configuration.
2 When bit 0 (SVS) of parameter No. 3111 is set to 0, "SERVO SETTING" and "SERVO TUNING" are not displayed. When bit 1 (SPS) of parameter No. 3111 is set to 0, "SPINDLE TUNING" is not displayed.

12.4.23.2 Parameter tuning screen (system setting) (15/19-inch display unit)

This screen enables the parameters related to the entire system configuration to be displayed and modified. The parameters can be initialized to the standard values (recommended by FANUC).

Display and setting

Procedure

1 Move the cursor to [SYSTEM SETTING] by pressing cursor key ↑ or ↓ on the parameter tuning menu screen described in Subsection, “Displaying the menu screen and selecting a menu item (15/19-inch display unit)”.

2 Press horizontal soft key [SELECT]. The screen display switches to the screen and horizontal soft keys shown below.

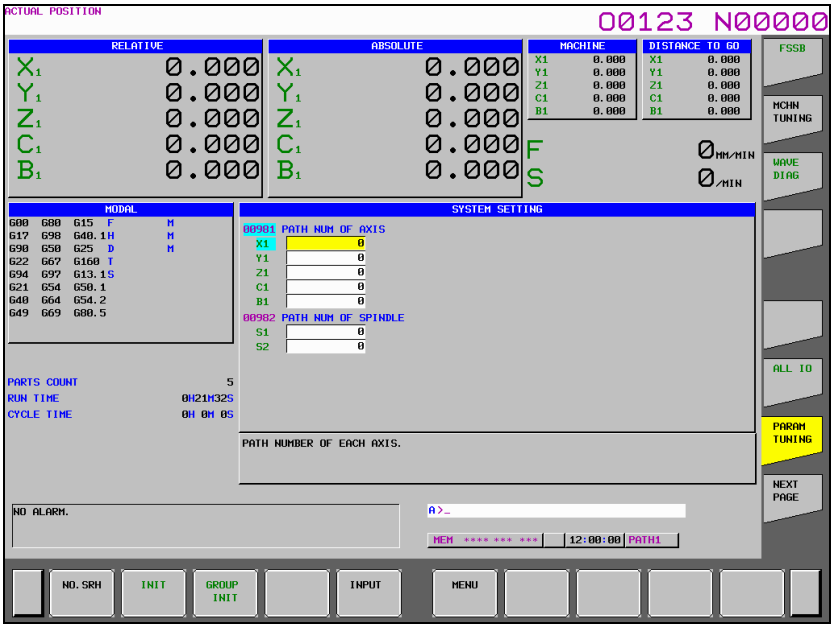


Fig. 12.4.23.2 (a) Parameter tuning screen (system setting) (15-inch display unit)

3 Move the cursor to a parameter number to be set or displayed, according to one of the methods below.

- Input the desired parameter number, then press horizontal soft key [NO.SRH].
- Move the cursor to the desired parameter number by pressing page key PAGE ↑ or PAGE ↓, and cursor keys ↑, ↓, ←, and/or →.

  A brief description of the parameter where the cursor is placed is provided at the bottom of the screen. However, no description is provided when the cursor is placed on multiple bits for bit parameters.

4 Input desired data then press the INPUT key on the MDI unit to set the parameter.

5 Press horizontal soft key [INIT]. The standard value (recommended by FANUC) for the item selected by the cursor is displayed in the key input buffer. Pressing horizontal soft key [EXEC] in this state initializes the item to the standard value.

6 Press horizontal soft key [GROUP INIT]. A message asking whether to set the group standard values is displayed on the screen. Pressing horizontal soft key [EXEC] in this state inputs all of the standard values of the group.

NOTE
1 If the cursor is placed on a parameter that has no standard value assigned, no standard value is input even when [INIT] is pressed.
2 When the cursor is placed on multiple bits for bit parameters, the multiple bits can be input simultaneously. When [INIT] is pressed in this state, the key input buffer displays the standard values for the bits where the cursor is placed. If a bit has no standard value assigned, "*" is displayed for the bit, and no value is input for the bit.
3 When [GROUP INIT] is pressed, those parameters that have no standard values assigned are not initialized.

12.4.23.3 Parameter tuning screen (axis setting) (15/19-inch display unit)

This screen enables the CNC parameters related to axes, coordinates, feedrate, and acceleration/deceleration to be displayed and set. The parameters displayed can be divided into four groups:

(Basic) group : The parameters related to basic settings are displayed.
(Coordinate) group : The parameters related to coordinates are displayed.
(Feedrate) group : The parameters related to feedrate are displayed.
(Acceleration/deceleration) group : The parameters related to acceleration/deceleration are displayed.

The parameters can be initialized to the standard values (recommended by FANUC). For the display and setting procedure, see the description of the parameter tuning screen (axis setting) in Subsection, “Parameter tuning screen (system setting) (15/19-inch display unit)”.

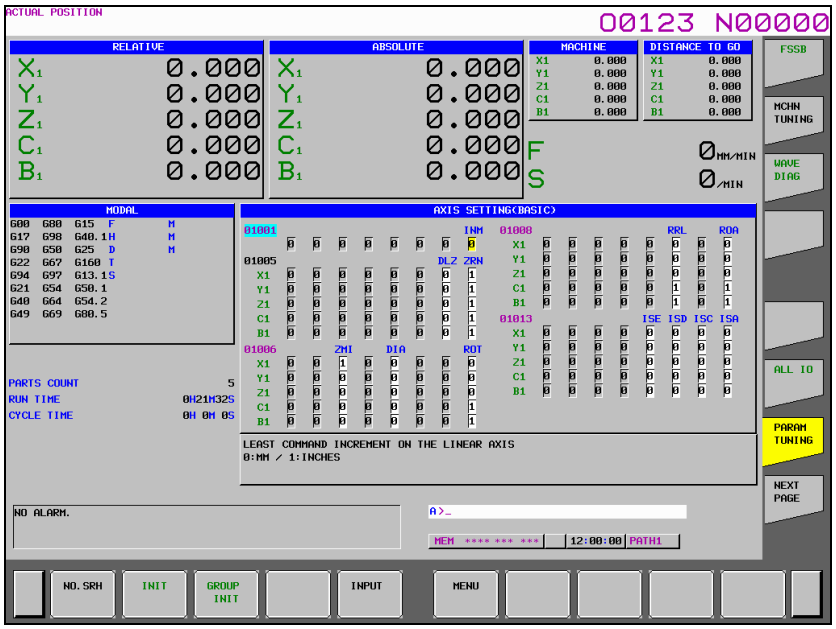


Fig. 12.4.23.3 (a) Parameter tuning screen (axis setting) (15-inch display unit)

12.4.23.4 Displaying and setting the FSSB servo amplifier setting screen (15/19-inch display unit)

From the parameter tuning screen, the FSSB servo amplifier setting screen can be displayed. For details of the FSSB servo amplifier setting screen, see the description of the FSSB servo amplifier setting screen in Subsection, “FSSB Setting” in the Connection Manual (Function) (B-64483EN-1).

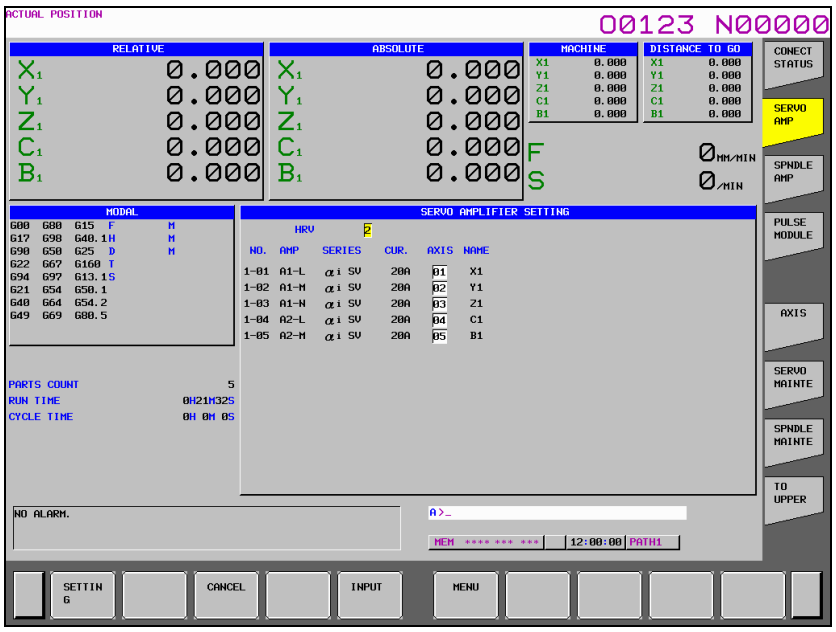


Fig. 12.4.23.4 (a) FSSB servo amplifier setting screen (15-inch display unit)

12.4.23.5 Displaying and setting the FSSB spindle amplifier setting screen (15/19-inch display unit)

From the parameter tuning screen, the FSSB spindle amplifier setting screen can be displayed. For details of the FSSB spindle amplifier setting screen, see the description of the FSSB spindle amplifier setting screen in Subsection, “FSSB Setting” in the Connection Manual (Function) (B-64483EN-1).

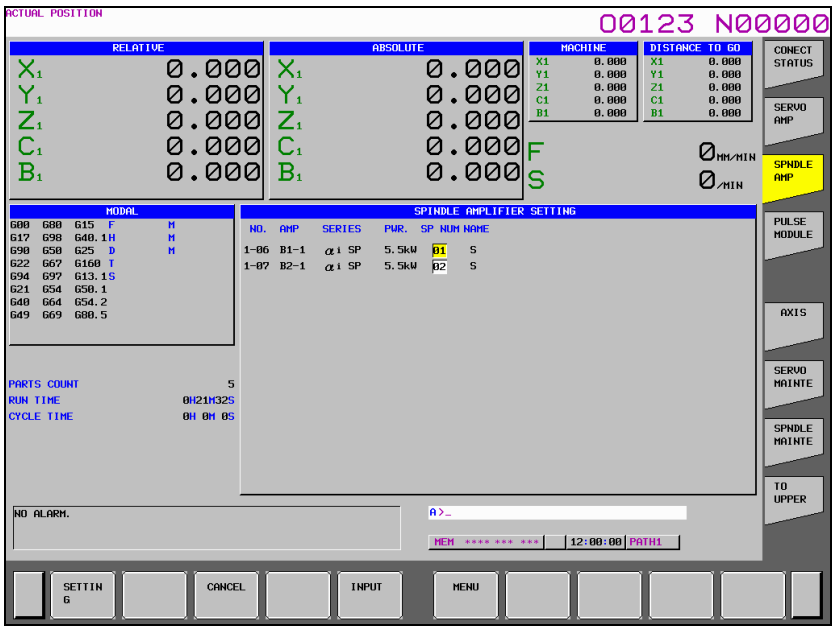


Fig. 12.4.23.5 (a) FSSB spindle amplifier setting screen (15-inch display unit)

12.4.23.6 Displaying and setting the FSSB axis setting screen (15/19-inch display unit)

From the parameter tuning screen, the FSSB axis setting screen can be displayed. For details of the FSSB axis setting screen, see the description of the FSSB axis setting screen in Subsection, “FSSB Setting” in the Connection Manual (Function) (B-64483EN-1).

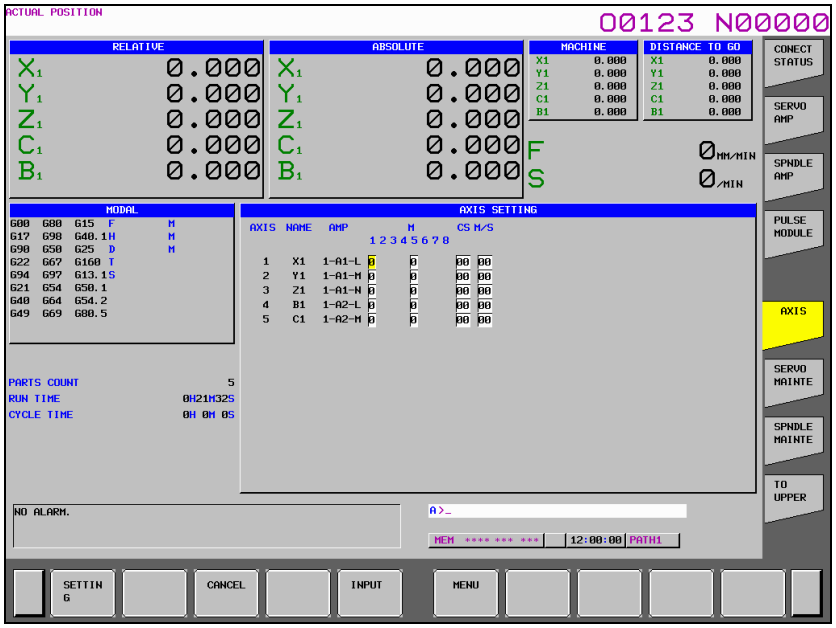


Fig. 12.4.23.6 (a) FSSB axis setting screen (15-inch display unit)

12.4.23.7 Displaying and setting the servo setting screen (15/19-inch display unit)

From the parameter tuning screen, the servo setting screen can be displayed. For details of the servo setting screen, see the Subsection, “Servo Parameters (15/19-inch Display Unit)”.

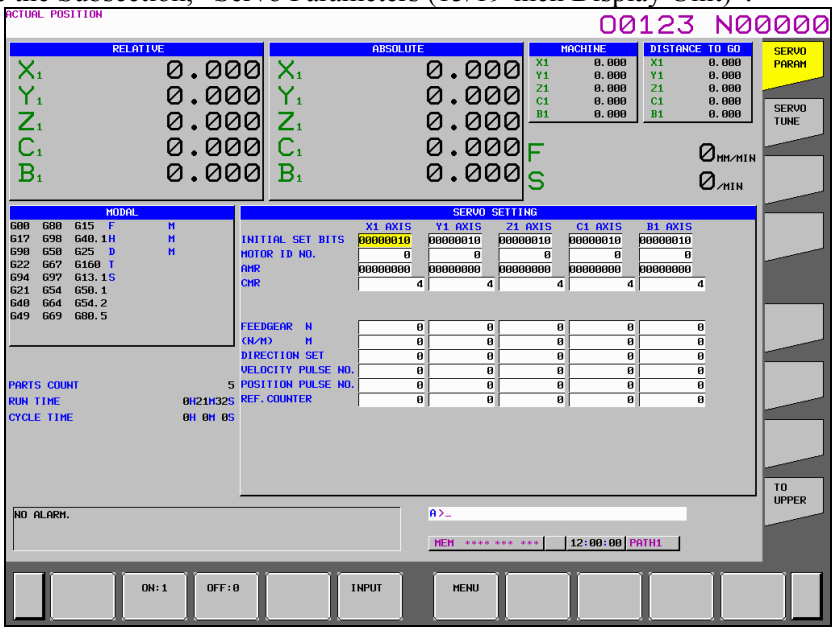


Fig. 12.4.23.7 (a) Servo setting screen (15-inch display unit)

12.4.23.8 Parameter tuning screen (spindle setting) (15/19-inch display unit)

The spindle-related parameters can be displayed and modified.
For the display and setting procedure, see the Subsection, “Parameter tuning screen (system setting) (15/19-inch display unit)”.

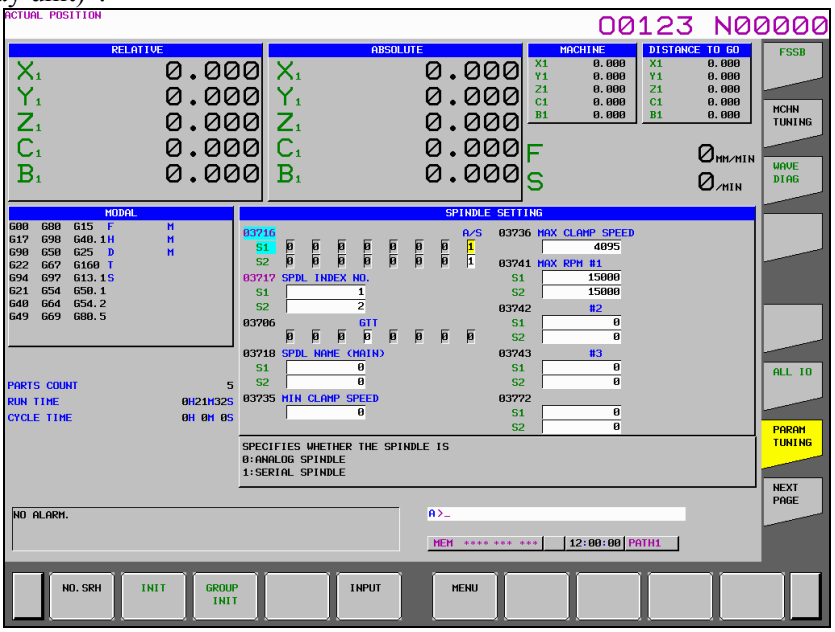


Fig. 12.4.23.8 (a) Parameter tuning screen (spindle setting) (15-inch display unit)

12.4.23.9 Parameter tuning screen (miscellaneous settings) (15/19-inch display unit)

The parameters related to the allowable number of M code digits and whether to display the servo setting and spindle tuning screens can be displayed and modified. Moreover, the parameters can be initialized to the standard values (recommended by FANUC).
For the display and setting procedure, see the Subsection, “Parameter tuning screen (system setting) (15/19-inch display unit)”.

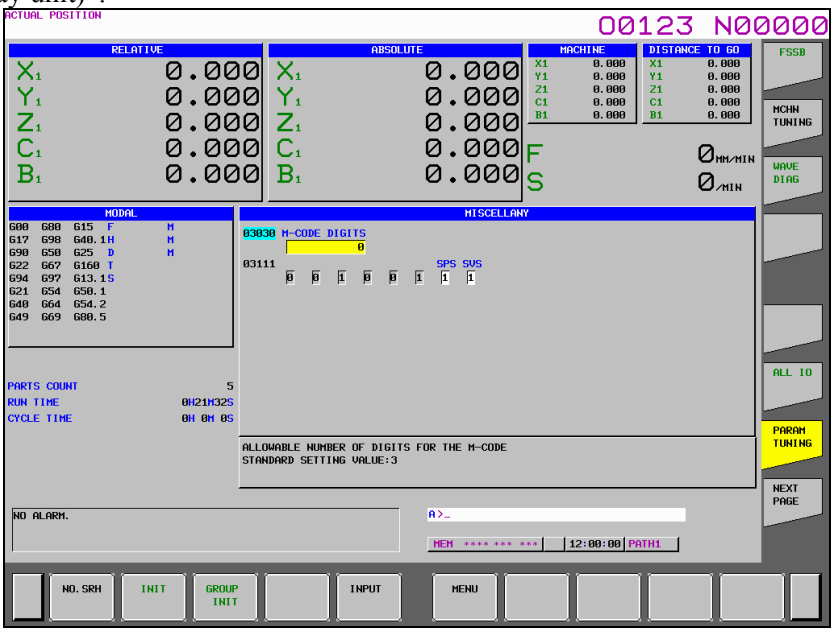


Fig. 12.4.23.9 (a) Parameter tuning screen (Miscellaneous setting) (15-inch display unit)

12.4.23.10 Displaying and setting the servo tuning screen (15/19-inch display unit)

From the parameter tuning screen, the servo tuning screen can be displayed. For details of the servo tuning screen, see the Subsection, “Servo Tuning (15/19-inch Display Unit)”.

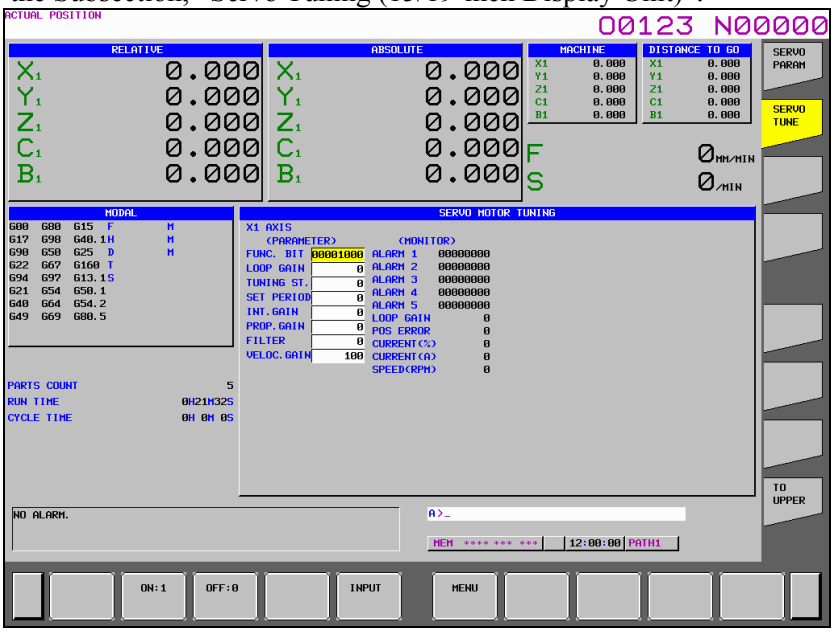


Fig. 12.4.23.10 (a) Servo tuning screen (15-inch display unit)

12.4.23.11 Displaying and setting the spindle tuning screen (15/19-inch display unit)

From the parameter tuning screen, the spindle tuning screen can be displayed. For details of the spindle tuning screen, see the Subsection, “Spindle Tuning (15/19-inch Display Unit)”.

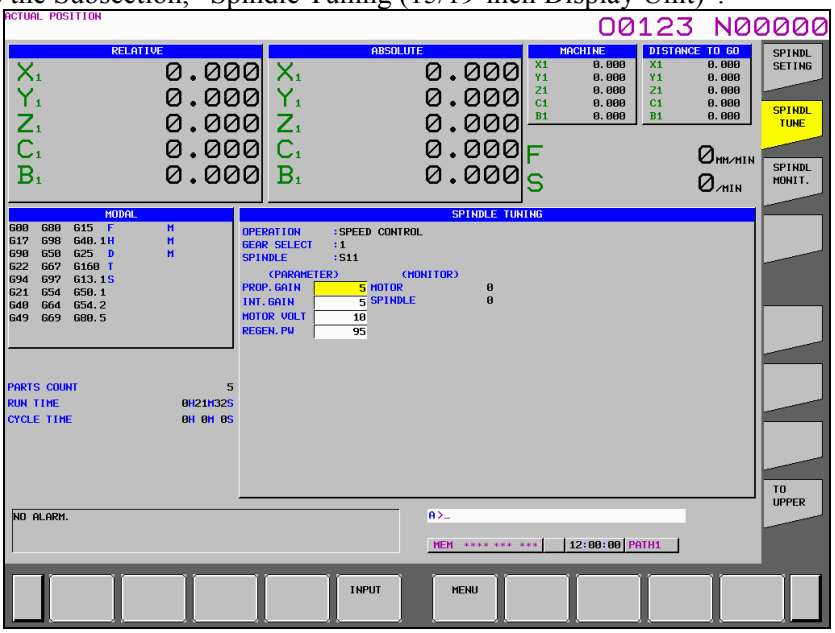


Fig. 12.4.23.11 (a) Spindle tuning screen (15-inch display unit)

12.4.23.12 Displaying and setting the machining parameter tuning screen (15/19-inch display unit)

From the parameter tuning screen, the machining parameter tuning screen can be displayed. For details of the machining parameter tuning screen, see the Subsection, “Machining Parameter Tuning (15/19-inch Display Unit)”.

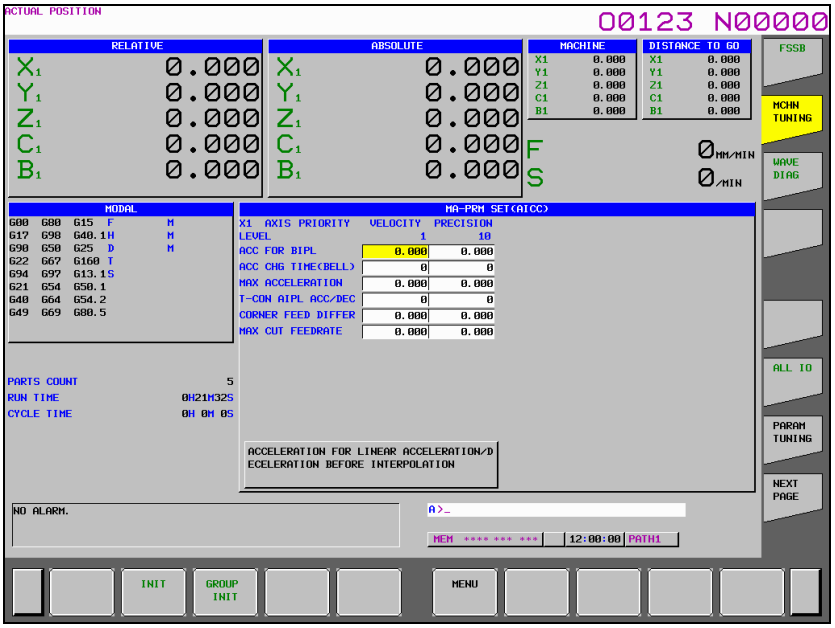


Fig. 12.4.23.12 (a) Machining parameter tuning screen (15-inch display unit)

Explanation

- Parameters displayed for parameter tuning

Table 12.4.23 (a) Parameters displayed for parameter tuning (1)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
SYSTEM SETTING	System setting	981		Sets the path of each axis.	
		982		Sets the path of each spindle.	
		983		Sets the T series/M series of each path. 0:T series / 1:M series	
		984#0	LCP	Sets the attribute of each path. 0:Normal / 1:Loader control	*1
		3021		Sets the G/F signal address of each axis.	*2
		3022		Sets the G/F signal address of each spindle.	*3
		3006#0	GDC	The deceleration signal for reference position return is: 0:X signal / 1:G signal	1
		3008#2	XSG	The signal address allocated to address X is: 0:Fixed / 1:Set by the parameter	1
		3013		Address allocated to the deceleration signal for reference position return	*4
		3014		Bit position allocated to the deceleration signal for reference position return	*5

*1 : The value 1 is set for the paths as many as the number of loader paths starting from the greatest path number. For path 1, the value 0 is set at all times.
Example) When the number of loader paths is 3 in a 10-path system:
The value 1 is set for paths 8 to 10. The value 0 is set for others.

*2 : When intra-path axis number ≤ 8, (path number - 1)*10+(intra-path axis number - 1)
When intra-path axis number ≥ 9, no standard value is available.
Example) When path 1 has 9 axes, and path 2 has 3 axes:
0,1,...,7,(none) for axes of path 1, 10,11,12 for axes of path 2

*3 : When intra-path spindle number ≤ 4, (path number - 1)*10+(intra-path spindle number - 1)
When intra-path spindle number ≥ 5, no standard value is available.
Example) When path 1 has 5 spindles, and path 2 has 1 spindle:
0,1,...,4,(none) for spindles of path 1, 10 for spindle of path 2

*4 : When (path number ≤ 3) and (intra-path axis number ≤ 8)
Axes of path 1: 9
Axes of path 2: 7
Axes of path 3: 10
Other axes: No standard value is available.
Example) When path 1 has 9 axes, and path 2 has 3 axes:
9,9,9,9,9,9,9,9,(none) for axes of path 1, 7,7,7 for axes of path 2

*5 : When (path number ≤ 3) and (intra-path axis number ≤ 8)
(Intra-path axis number - 1)
Other axes: No standard value is available.
Example) When path 1 has 9 axes, and path 2 has 3 axes:
0,1,...,7,(none) for axes of path 1, 0,1,2 for axes of path 2

Table 12.4.23 (b) Parameters displayed for parameter tuning (2)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
SPINDLE SETTING	Spindle setting	3716#0	A/S	Sets the type of spindle motor: 0:Analaog / 1:Serial.	
		3717		Sets a motor number to be assigned to each spindle.	
		3706#4	GTT	Specifies a spindle gear selection method. 0:M type / 1:T type	
		3718		Sets a suffix to be added to spindle speed display on a screen such as the position display screen.	
		3735		Sets the minimum clamp speed of the spindle motor.	
		3736		Sets the maximum clamp speed of the spindle motor.	
		3741		Sets the maximum spindle speed for gear 1.	
		3742		Sets the maximum spindle speed for gear 2.	
		3743		Sets the maximum spindle speed for gear 3.	
		3744		Sets the maximum spindle speed for gear 4.	
		3772		Sets the maximum speed of the spindle. When 0 is set, the speed is not clamped.	
		4133		Sets the motor model code of the serial spindle. (This setting is not required for an analog spindle.)	
		4019#7	***	The parameters for the serial spindle are: 0:Not automatically set/1:Automatically set (This setting is not required for an analog spindle.)	

Table 12.4.23 (c) Parameters displayed for parameter tuning (3)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
AXIS SETTING	Basic	1001#0	INM	Least command increment on linear axes: 0:Metric (millimeter machine) / 1:Inch (inch machine)	
		1005#0	ZRNx	When an automatic operation (other than G28) is executed before reference position return: 0:Alarm is issued (PS0224) / 1:No alarm is issued.	0
		1005#1	DLZx	Reference position return without dogs: 0:Disabled / 1:Enabled	
		1006#0	ROTx	Setting of linear axes or rotary axis: 0:Linear axis / 1:Rotation axis	
		1006#3	DIAx	Setting of the amount of travel: 0:Radius specification / 1: Diameter specification	
		1006#5	ZMIx	Reference position return direction: 0:Plus direction / 1:Minus direction	
		1008#0	ROAx	Rotation axis roll-over function: 0:Disabled / 1:Enabled	1
		1008#2	RRLx	With the amount of travel per revolution, relative coordinates are: 0:Not rounded / 1:Rounded	1
		1013#0	ISAx	Sets the least input increment and least command increment: 0:IS-B / 1:IS-A	
		1013#1	ISCx	Sets the least input increment and least command increment: 0:IS-B / 1:IS-C	
		1013#2	ISDx	Sets the least input increment and least command increment: 0:IS-B / 1:IS-D	
		1013#3	ISEx	Sets the least input increment and least command increment: 0:IS-B / 1:IS-E	
		1020		Program axis name	*1
		1022		Sets each axis in the basic coordinate system.	*2
		1023		Servo axis number	
		1815#1	OPTx	A separate pulse coder is: 0:Not used / 1:Used	
		1815#4	APZx	The correspondence between machine positions and absolute-position detector positions is: 0:Not established / 1:Established	
		1815#5	APCx	The position detector used is: 0:Other than an absolute-position detector / 1:Absolute-position detector	
		1825		Servo loop gain	
		1826		Effective area	
		1828		Positional deviation limit during travel	
		1829		Positional deviation limit during stop	500

*1 : For the M series: 88(X),89(Y),90(Z) sequentially starting with the first axis (no standard value for the fourth axis and up)
For the T series: 88(X),90(Z) (no standard value for the third axis and up)

*2 : For the M series: 1,2,3 sequentially starting with the first axis (no standard value for the fourth axis and up)
For the T series: 1,3 (no standard value for the third axis and up)

Table 12.4.23 (d) Parameters displayed for parameter tuning (4)

Menu	Group	Parameter No.	Name	Brief description	Standard setting
AXIS SETTING	Coordinate	1240		Machine coordinate of the first reference position	
		1241		Machine coordinate of the second reference position	
		1260		Amount of travel per revolution of a rotary axis	360.000
		1320		Coordinate of the boundary of stored stroke check 1 in the positive direction	
		1321		Coordinate of the boundary of stored stroke check 1 in the negative direction	
	Feedrate	1401#6	RDR	For a rapid traverse command, dry run is: 0:Disabled / 1:Enabled	0
		1410		Dry run feedrate	
		1420		Rapid traverse rate	
		1421		Rapid traverse override F0 rate	
		1423		Jog feedrate	
		1424		Manual rapid traverse rate	
		1425		FL feedrate for reference position return	
		1428		Reference position return feedrate	
		1430		Maximum cutting feedrate	
	Accele-ra tion/ decele-ration	1610#0	CTLx	Acceleration/deceleration in cutting feed or dry run during cutting feed: 0: Exponential acceleration/deceleration is applied / 1: Linear acceleration/deceleration after interpolation is applied	
		1610#4	JGLx	Acceleration/deceleration for jog feed is: 0: Exponential acceleration/deceleration / 1: Same as acceleration/deceleration for cutting feed (The settings of bit 1 (CTBx) and bit 0 (CTLx) of parameter No. 1610 are followed.)	
		1620		Time constant for linear acceleration/deceleration for rapid traverse	
		1622		Time constant for acceleration/deceleration for cutting feed	
		1623		FL feedrate for acceleration/deceleration after interpolation for cutting feed	
		1624		Time constant for acceleration/deceleration for jog feed	
		1625		FL feedrate for exponential acceleration/deceleration for jog feed	
MISCELLANY	MISC	3030		Allowable number of digits of an M code	3
		3111#0	SVS	The servo setting screen is: 0:Not displayed / 1:Displayed	1
		3111#1	SPS	The spindle setting screen is: 0:Not displayed / 1:Displayed	1

12.4.24 Periodic Maintenance Screen (15/19-inch Display Unit)

Periodic maintenance screens are used for managing consumables (such as the backlight of a LCD unit and backup batteries). By setting the name of a consumable, its life time, and the method for counting down the life time, the life of the consumable can be counted down in the appropriate method set for the consumable, and the remaining life time can be displayed.
With these screens, the user can easily manage consumables that require periodic replacement.
The total number of item that can be managed is 10 in all path.

Explanation

There are four periodic maintenance screens: the status screen, the setting screen, the machine menu screen, and the NC menu screen.

Status screen : Item names, remaining times, and count statuses are displayed, and item names are set.
Setting screen : Life times, remaining times, count types (count-down method), and path number (in case of multi-path system) are set.
Machine menu screen : The names of consumables in the machine can be registered.
NC menu screen : The names of consumables in the NC are already registered.

Using periodic maintenance screens

<1> Referencing a periodic maintenance screen
Display the status screen. For the meanings of items on the status screen, see Item, "**Status screen**".
<2> Adding the name of a new consumable item to a periodic maintenance screen or editing an existing consumable item on the screen
The name of a consumable item can be added or edited on the machine menu screen. For details, see Item, "**Machine menu screen**".
<3> Adding or editing a life time, remaining time, and a method of counting down the remaining time for a consumable on a periodic maintenance screen
A life time and remaining time can be added or edited on the setting screen. For details, see Item, "**Setting screen**".
<4> Newly displaying the item name and remaining time of a consumable on a periodic maintenance screen.
1 Setting an item name
Select the item name of a consumable to be displayed from the machine menu screen or NC menu screen, or enter the name using the MDI keys. For the procedure, see Item, "**Item name**" in "**Status screen**".
2 Setting a life time, remaining time, and count type
Select the life time, remaining time, and count type of a consumable to be displayed from the setting screen. For the procedure, see Item, "**Remaining time**" in "**Status screen**".

Procedure for displaying a periodic maintenance screen

1 Press function key 

.
2 Press vertical soft key [NEXT PAGE] several times to display [PERIOD MAINTE].
3 Press vertical soft key [PERIOD MAINTE] to display the periodic maintenance screen.

Status screen

When vertical soft key [STATUS] is pressed, the status screen is displayed. The status screen shows the item names, count statuses, and remaining times of managed consumables.

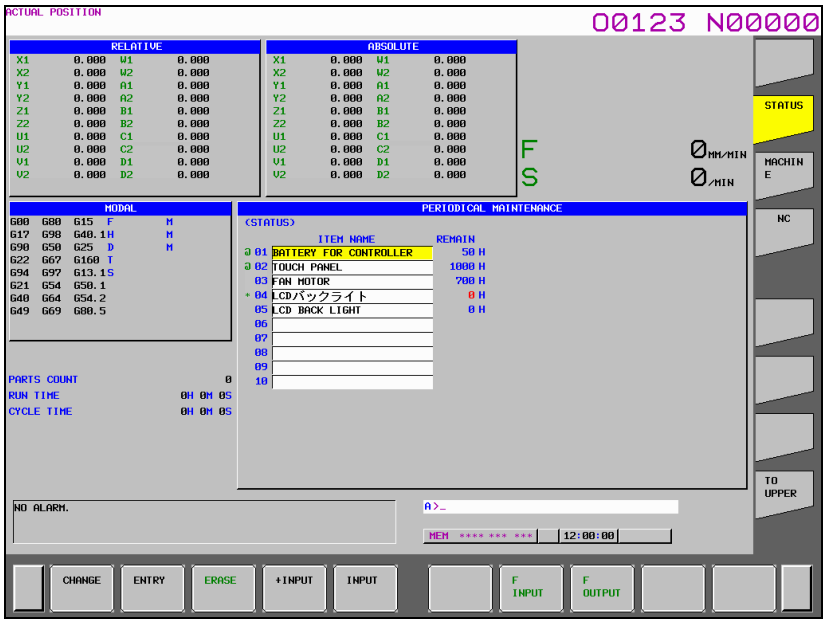


Fig. 12.4.24 (a) Status screen (15-inch display unit)

- Item name

As the item name, set the name of a consumable to be managed by periodic maintenance. To set an item name, select a name from the machine menu screen or NC menu screen, or directly enter the name using the MDI keys.

Setting an item name from a menu screen

1 On the status screen, move the cursor to the name of the item to be set, and press horizontal soft key [ENTRY].
2 Press vertical soft key [MACHINE] or [NC] to display the machine menu screen or NC menu screen.
3 Move the cursor to an item name registered on the menu screen, and press horizontal soft key [SELECT] then [EXEC] successively.
4 The screen display returns to the status screen, and the item name selected on the menu screen is added to the status screen.

Initially, there is no item name set on the machine menu screen, so item names must be registered in advance. For the registration method, see the description of the procedure for registering item names on the machine menu screen.

Setting an item name using the MDI keys

1 Type the alphanumeric characters to be input, then press horizontal soft key [INPUT].
2 The entered item name is registered on the status screen.

When horizontal soft key [+INPUT] is pressed instead of horizontal soft key [INPUT], the entered characters can be added to an existing item name.
When typing 2-byte characters, type "*" before and after the character codes. The 2-byte character codes must conform to FANUC codes. (See Appendix, "FANUC 2-BYTE CHARACTER CODE TABLE".)
An item name to be registered must be up to 24 characters long if it consists of alphanumeric characters only; or it must be up to 12 characters long if it consists of 2-byte characters only.
Example: To register "LCD バックライト", enter the following:

>LCD*110E10F410CC114010B610FE*_

NOTE
1 An asterisk "*" is used as a control code, so it cannot be used in item names. In addition, characters "[", "]", "(", and ")" cannot be used in item names.
2 When an item name consisting of both alphanumeric characters and 2-byte characters is registered, the warning "DATA OUT OF RANGE" may be output.

Deleting an item name
To delete a registered item name, move the cursor to the item name, press horizontal soft key [ERASE], then press horizontal soft key [EXEC].
When an item name is deleted, its life time, remaining time, and count type are deleted at the same time.

- Remaining time

As the remaining time, the period of time left until the time for replacement is reached by count down operation is displayed.
If the percentage of the remaining time to the life time has reached to the value (%) specified by parameter No. 8911, or smaller, the remaining time is displayed in red.
Even after the life time has expired, count down operation continues.

Setting the remaining time
1 On the status screen, place the cursor on an item for which the remaining time is to be set (the item name must have been set in advance).
2 Press horizontal soft key [CHANGE].
3 The screen display changes to the setting screen.
4 Set the life time, remaining time, and count type. For the setting method and other information, see Item, "**Setting screen**".

NOTE
On the status screen, the remaining time and life time cannot be set.
These items must be set on the setting screen.

- Count status

The count status is indicated on the left side of the item number as follows:

Indication	Count status
Blank	Counting stopped
@	Counting in progress
*	Life expired

Setting screen

On the setting screen, the life time, remaining time, and count type of a managed consumable are set.

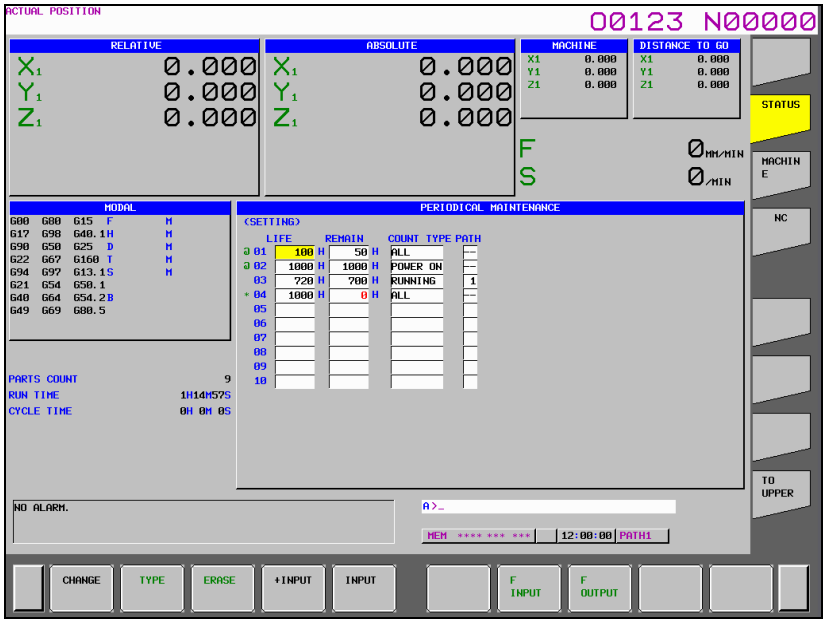


Fig. 12.4.24 (b) Setting screen (15-inch display unit)

Display procedure

1 Press horizontal soft key [CHANGE].

- Life time

Set the life time of a consumable.

Move the cursor to an existing item, type a life time, then press horizontal soft key [INPUT] (or the 

key). The life time is then set, and the same value is set also as the remaining time.
At this time, "------" is indicated in the count type field.
When horizontal soft key [+INPUT] is pressed, the entered value can be added to the life time already set. The same value as the added value is added also to the remaining time.
A value ranging from 0 to 65535 (in hours) can be set.

NOTE

1 If a setting operation is attempted when the item name is not registered, the warning "EDIT REJECTED" is issued.
2 If a value exceeding the valid range is entered, the warning "DATA IS OUT OF RANGE" is issued.
3 If horizontal soft keys [ERASE] and [TYPE] is pressed, the warning "EDIT REJECTED" is issued.

- Remaining time

The period of time left until the time for replacement is reached by count down operation is indicated.
If the percentage of the remaining time to the life time has reached the value (%) specified by parameter No. 8911, or smaller, the remaining time is displayed in red.
Even after the life time has expired, count down operation continues.
Move the cursor to the remaining time of a target registered number, type a remaining time, then press horizontal soft key [INPUT] (or the 

key). The remaining time is then set.

When horizontal soft key [+INPUT] is pressed, the entered value can be added to the remaining time already set.

A value ranging from 0 to (life time) can be set.
When horizontal soft key [ERASE] then horizontal soft key [EXEC] are pressed, the same value as the life time is set.

NOTE
1 If a setting operation is attempted when the item name or life time is not registered, the warning “EDIT REJECTED” is issued.
2 When a value exceeding the valid range is entered, the warning “DATA IS OUT OF RANGE” is issued.
3 If horizontal soft key [TYPE] is pressed, the warning “EDIT REJECTED” is issued.

- Count type

As the count type, select the way of counting.
Place the cursor on the count type of a target registration number, then press horizontal soft key [TYPE]. Count types are displayed as horizontal soft keys as shown below. Select one of these horizontal soft keys, then press horizontal soft key [EXEC].

Horizontal soft key	Meaning	Indication
[NO CNT]	Counting is not performed (stopped)	———————
[ALL]	Counting is performed at all times	All times
[POWER ON]	Counting is performed when power is on.	When power is on
[RUN]	Counting is performed when operation is in progress.	When operation is in progress
[CUT]	Counting is performed when cutting is being performed.	When cutting is being performed

NOTE
1 If a setting operation is attempted when the item name or life time is not registered, the warning “EDIT REJECTED” is issued.
2 Horizontal soft keys [INPUT] and [+INPUT] have no effect.
3 When counting is performed at all times, a 24-hour error is generated in leap year.
4 If horizontal soft key [ERASE] is pressed, the warning “EDIT REJECTED” is issued.

- Path number

When [RUN] or [CUT] is set to the count type in multi path, path number can be set.
Make the cursor to the count type of a target registration number, then input path number and press soft key [INPUT].
The setting range is 0-(max path number).
When the specified path is operating or cutting, counting is performed.
If 0 is set, when either of path is operating or cutting, counting is perfomed.

NOTE
1 If [NO CNT], [ALL] or [POWER ON] is set to count type, “--” is displayed. In this case, if the setting is operated, the warning “EDIT REJECTED” is issued.
2 If the setting is beyond the valid data range, the warning “DATA IS OUT OF RANGE” is issued.
3 If soft key [TYPE] is pressed, the warning “EDIT REJECTED” is issued.
4 If soft key [ERASE] is pressed, the warning “EDIT REJECTED” is issued. The soft key [+INPUT] is invalid.

Registration from a program

The item name, life time, remaining time, count type and path number can be registered in the status screen and the setting screen by executing the program of the following format.

Format

G10 L60 Px [n] Aa Rr Qpt

X : Registration number

n : Item name, format: [alphanumeric-characters*2-byte-characters*alphanumeric-characters]

a : life time

r : remaining time

p : path number

t : count type

The meaning of the value of the count type is as follow.

0 : NO COUNT

1 : ALL TIMES

2 : POWER ON

3 : RUNNING

4 : CUTTING

The range of the path number is 0 – (max path number).

EX.)

Q24 : The path number is 2, and the count type is “CUTTING”

Q103 : The path number is 10, and the count type is “RUNNING”

Machine menu screen

On the machine menu screen, the names of consumables of the machine are registered. From this screen, item names can be added to the status screen. For the method of addition to the status screen, see the description of the status screen.

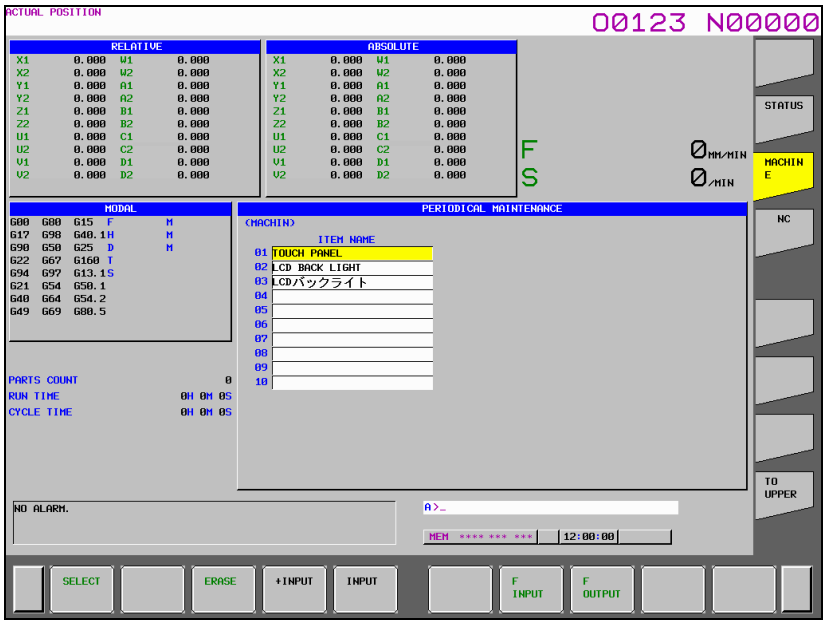


Fig. 12.4.24 (c) Machine menu screen (15-inch display unit)

- Displaying the screen

1 When the status screen is displayed, press vertical soft key [MACHINE].

On the machine menu screen, item names can be registered using one of the following two methods:

- Registration from a program
- Registration using the MDI keys

- Registration from a program

By executing a program having the following format, an item name can be registered in the menu of the machine:

Format

G10 L61 Px [n]

X Registration number

n Item name, format: [alphanumeric-characters*2-byte-characters*alphanumeric-characters]

- Registration using the MDI keys

An item name can be registered on the menu screen of the machine by typing the item name in the format shown below on the machine menu screen, then pressing horizontal soft key [INPUT] (or the 

key).

When horizontal soft key [+INPUT] is pressed, the typed characters can be added to an already registered item name.

Format

Alphanumeric-characters*2-byte-characters*alphanumeric-characters

Two-byte codes must conform to FANUC codes. (See Appendix, "FANUC 2-BYTE CHARACTER CODE TABLE".)
When typing 2-byte characters, type an asterisk "*" before and after the character codes. An item name to be registered must be up to 24 characters long if it consists of alphanumeric characters only; or it must be up to 12 characters long if it consists of 2-byte characters only.

Example:

To register "LCD バックライト", enter the following:
>LCD*110E10F410CC114010B610FE*_

NOTE

1 An asterisk "*" is used as a control code, so it cannot be used in item names. In addition, characters "[", "]", "(", and ")" cannot be used in item names.
2 When an item name consisting of both alphanumeric characters and 2-byte characters is registered, the warning “DATA IS OUT OF RANGE” may be output.
3 When a blank item name is selected on the machine screen, the warning “EDIT REJECTED” is issued.

To delete a registered item name, move the cursor to the item name, press horizontal soft key [ERASE], then press horizontal soft key [EXEC].

NC menu screen

From this screen, an item name can be registered on the status screen. For the method of registration to the status screen, see the description of the status screen.

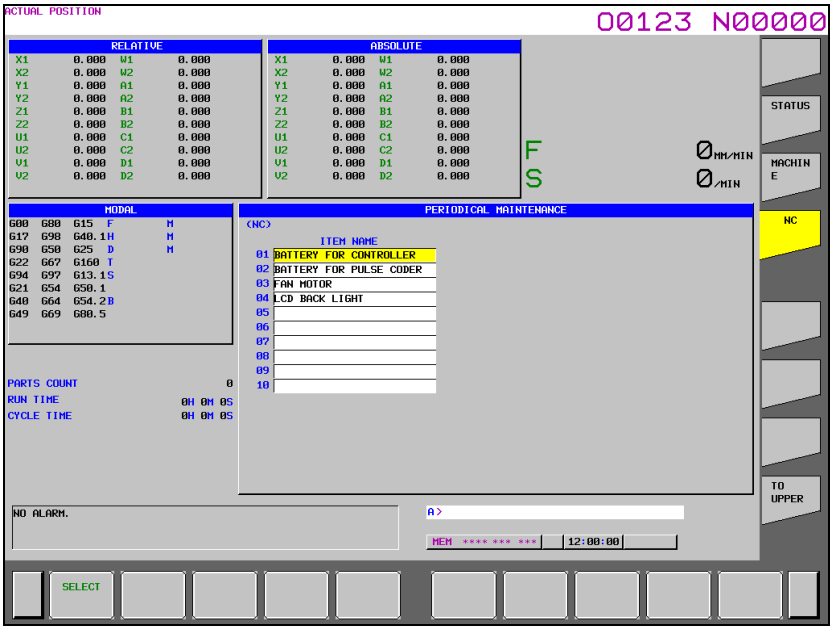


Fig. 12.4.24 (d) NC menu screen (15-inch display unit)

- Displaying the screen

1 When the status screen is displayed, press vertical soft key [NC].

NOTE

On the NC screen, the registration, deletion, and I/O of item names cannot be performed.
When a blank item name is selected, a blank is set.

12.4.25 System Configuration Screen (15/19-inch Display Unit)

The system configuration screen provides information about the types of installed hardware and software.

Procedure for displaying the screen

Procedure

1 Press the function key SYSTEM, then press vertical soft key [NEXT PAGE] several times until vertical soft key [SYSTEM] is displayed.
2 Press vertical soft key [SYSTEM]. The system configuration screen is displayed.

There are two types of system configuration screens: the hardware configuration screen and the software configuration screen. The screen display can switch between these screens by using PAGE ↑ and PAGE ↓.

When vertical soft key [SERVO INFO] or [SPINDLE INFO] is pressed, information about the connected servo system or spindles is displayed.

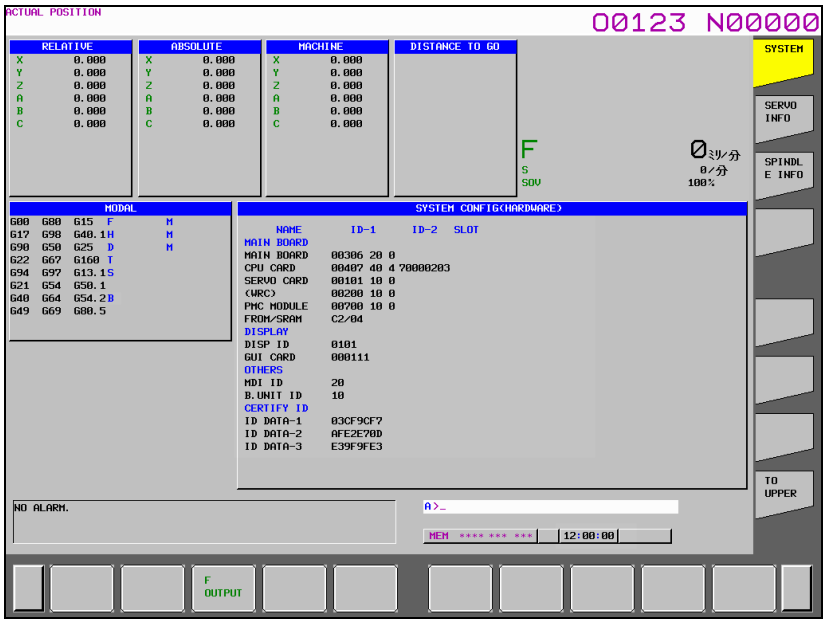


Fig. 12.4.25 (a) System configuration screen (15-inch display unit)

Hardware configuration screen

This screen shows the names and IDs of the hardware used by the NC.

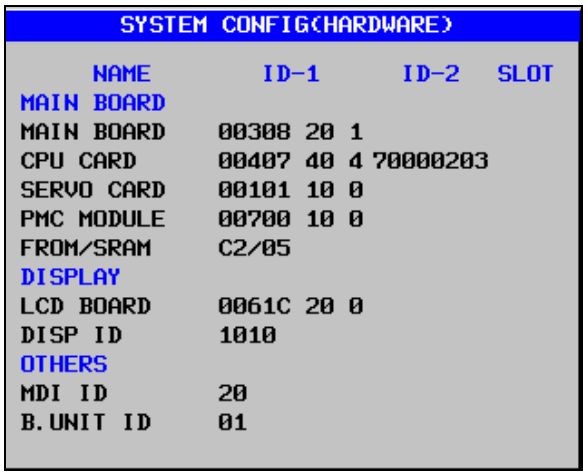


Fig. 12.4.25 (b) Hardware configuration screen

Software configuration screen

This screen shows the names and series/editions of the software used by the NC.

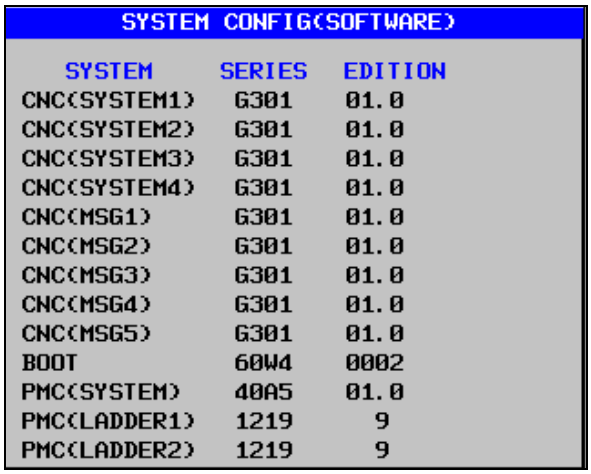


Fig. 12.4.25 (c) Software configuration screen

Servo information screen

When a servo system is connected to the NC, ID information of the connected servo devices (servo motors and servo amplifiers) can be displayed on the NC.

Displaying the screen

1 When the system configuration screen is displayed, press vertical soft key [SERVO INFO].
2 The servo information screen is displayed.

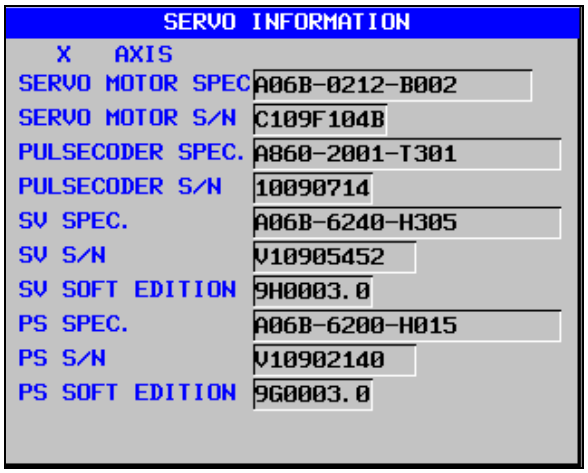


Fig. 12.4.25 (d) Servo information screen

Spindle information screen

When a spindle system is connected to the NC, the ID information of the connected spindle devices (spindle motors and spindle amplifiers) can be displayed on the NC.

Displaying the screen

1 When the system configuration screen is displayed, press vertical soft key [SPINDLE INFO].
2 The spindle information screen is displayed.

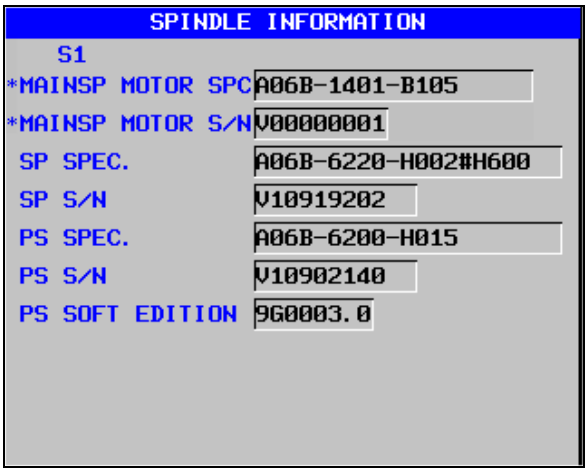


Fig. 12.4.25 (e) Spindle information screen

12.4.26 Power Consumption Monitoring Screen (15/19-inch Display Unit)

The electric power data of servo axis and spindle axis consumption and regeneration can be displayed.

Display of power consumption monitoring screen

1 Press function key SYSTEM.
2 Press continuous menu key ▷ several times until soft key [POWER MONIT] appears.
3 Press soft key [POWER MONIT].

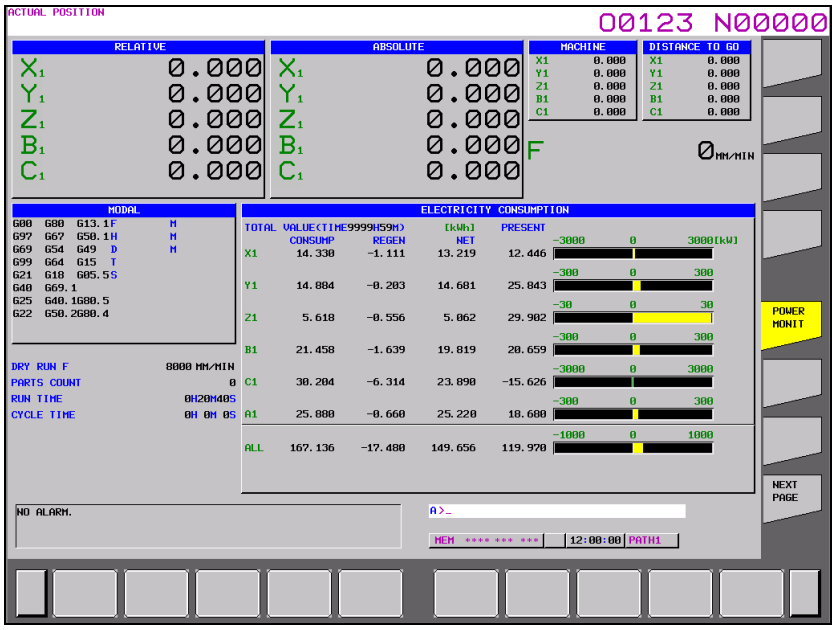


Fig. 12.4.26 Power consumption monitoring screen for 15” display unit

Operation of power consumption monitoring screen

Switch of display axis

When information on all axes is not displayed, the page is switched with the MDI key <PageUp> or <PageDown>.

Explanation

TIME

Integrating time of power consumption is displayed.

Axis name

Axis name of servo and spindle is displayed. "ALL" means the total of all servo and spindle axes.

CONSUMP

Integral power consumption is displayed.

REGEN

Integral power regeneration is displayed.

NET

Integral net amount of power consumption is displayed.
The net amount of power consumption = power consumption – power regeneration.

PRESENT

Present net power consumption is displayed. A negative value is displayed about the net power consumption while the power is regenerating like the axis is decelerating, etc.

Bar-graph of PRESENT

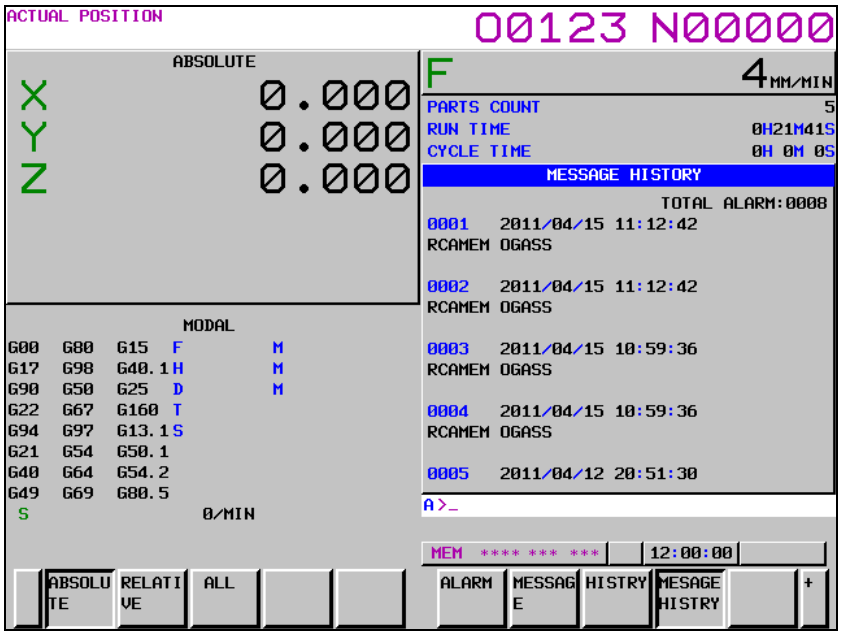
Present net power consumption is displayed as a bar-graph.
The scale of bar-graph can be specified in the parameter (No.11392 in case of the servo, No.11393 in case of the spindle).
The scale of the entire power consumption bar-graph can be specified in the parameter (No.11371).
The bar expands from the center to right while consuming the electric power. The bar expands from the center to left while regenerating the electric power.

12.5 SCREENS DISPLAYED BY FUNCTION KEY

By pressing the function key , data such as alarms, alarm history, external operator message, and external operator message history data can be displayed.
For details of alarms and alarm history, see Sections, “ALARM DISPLAY” and “ALARM HISTORY DISPLAY”. Refer to the machine tool builder's manual for the external message display.

12.5.1 External Operator Message History

External operator message can be stored as history. And, stored history can be seen on the external operator message history screen.



Screen display

To display the external operator message history screen, set bit 2 (OMH) of parameter No. 3112 to 1.

Procedure

1 Press function key .
2 Press soft key [MESSAGE HISTRY]. The external operator message history screen is displayed.
3 The screen display can be changed to the previous page and the next page by using page keys and .

Erasing history data from the external operator message history screen

Procedure

1 lay the external operator message history screen.
2 Press soft key [(OPRT)].
3 Press soft key [CLEAR]. The external operator message history data is then erased.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3112						OMH		

[Input type] Parameter input
[Data type] Bit

#2 OMH The external operator message history screen is:
0: Not displayed.
1: Displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3113	MS1	MS0						HMC

[Input type] Parameter input
[Data type] Bit

#0 HMC The contents of the external operator message history:
0: Cannot be erased.
1: Can be erased.

> **NOTE**
> This parameter is valid when bit 3 (SOH) of parameter No. 11354 is set to 1.

#6 MS0
#7 MS1 Set the combination of the number of characters and the number of messages to be preserved in the external operator message history.

Parameter		Maximum number of characters	Number of messages
MS0=0	MS1=0	255	8
MS0=1	MS1=0	200	10
MS0=0	MS1=1	100	18
MS0=1	MS1=1	50	32

> **NOTE**
> 1 Although up to 255 characters can be specified for each external operator message, you can use the combination of bits 6 (MS0) and 7 (MS1) of parameter No. 3113 to limit the number of characters and select the number of messages to be preserved in the external operator message history.
> 2 The settings of bits 6 (MS0) and 7 (MS1) of parameter No. 3113 take effect the next time the power is turned on. The external operator message history is erased at that time.
> 3 Even though you change the settings of bits 6 (MS0) and 7 (MS1) of parameter No. 3113, the alarm PW0000, "POWER MUST BE OFF" is not issued. You must however turn on the power again before the new settings can take effect.

NOTE
4 If text (such as single-byte katakana or kanji characters) is entered in character code, the number of characters recorded in the external operator message history may be smaller than the maximum number of characters set by bits 6 (MS0) and 7 (MS1) of parameter No. 3113.

	#7	#6	#5	#4	#3	#2	#1	#0
3196		HOM						

[Input type] Parameter input
[Data type] Bit

#6 HOM A history of external operator messages and macro messages ((#3006) is:
0: Recorded.
1: Not recorded.

	#7	#6	#5	#4	#3	#2	#1	#0
11354					SOH	SAH		

[Input type] Parameter input
[Data type] Bit

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#2 SAH When the storage capacity for history data is exceeded due to non-alarm history, alarm history will be:
0: Erased.
1: Erased, except the most recent 50 items of history data.

#3 SOH When the storage capacity for history data is exceeded due to data other than external operator message history, external operator message history will be:
0: Erased.
1: Retained.

NOTE
1 The settings of bit 2 (SAH) of parameter No. 11354 and bit 3 (SOH) of parameter No. 11354 will be effective the next time the power is turned on. At this time, all history data (operation history, alarm history, and external operator message history) will be erased.
2 With the settings of bit 2 (SAH) of parameter No. 11354 and bit 3 (SOH) of parameter No. 11354, the number of history data items that can be retained varies. The number of history data items that can be recorded as follows:
SAH=0, SOH=0 . . . Approx. 8000 items
SAH=1, SOH=0 . . . Approx. 7400 items
SAH=0, SOH=1 . . . Approx. 7500 items
SAH=1, SOH=1 . . . Approx. 6900 items
(*) The numbers of items above are those if only key operation history is recorded.

12.6 SWITCHING BETWEEN MULTI-PATH DISPLAY AND SINGLE-PATH DISPLAY FUNCTION

Overview

In a multi-path system, the screen can be switched between simultaneous multi-path display and single-path display by soft key operation.
This function allows easy switching between the single-path display which contains much information and the simultaneous multi-path display which enables multiple path information to be checked at a time.

Details

The screen can be switched between the simultaneous multi-path display and single-path display by soft key operation.
In the single-path display, the screen for the path selected by the path selection signal is displayed.
In the simultaneous multi-path display, the screen for the simultaneous display path group including the path selected by the path selection signal is displayed.
This function is enabled with the following screens which can display information of multiple paths simultaneously when bit 3 (MTS) of parameter No. 11355.

- Actual position display screen
  (Absolute position display screen, relative position display screen, overall position display screen)
- Program check screen
- Tool path graphic screen
- Program edit screen

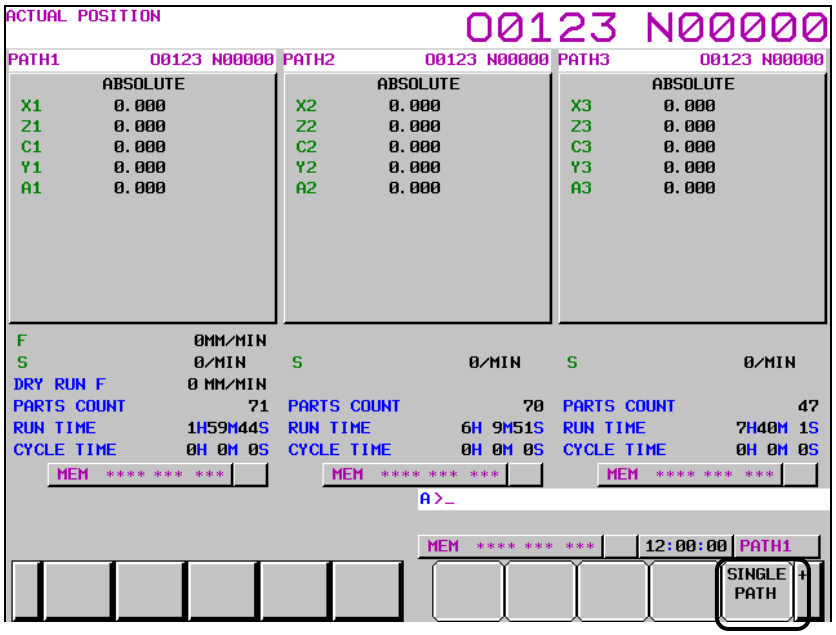
NOTE
For the setting of the simultaneous multi-path display, refer to the description of parameters Nos. 13131 and 13132 in the Parameter Manual (B-64490EN).

Procedure (8.4/10.4-inch display unit)

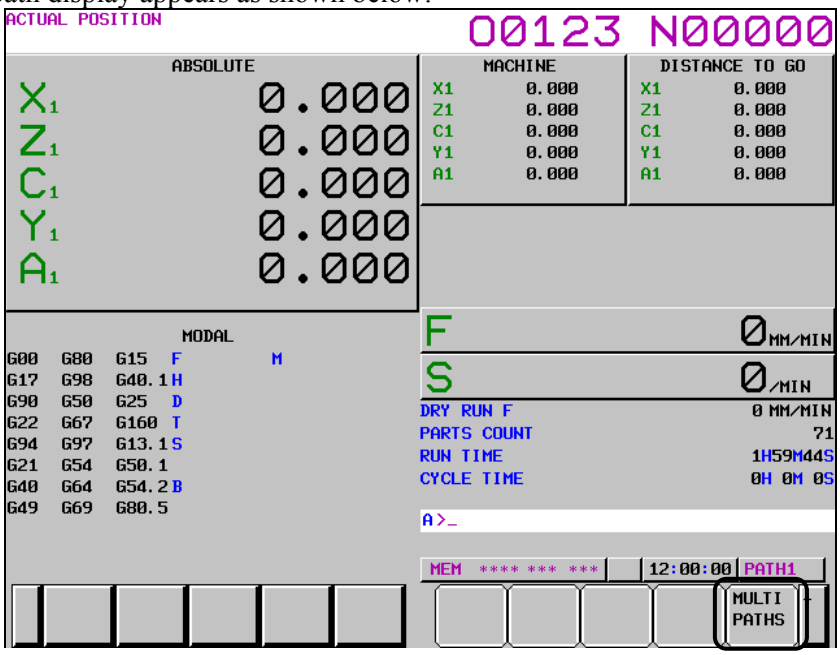
The procedure is explained using the current position display screen as an example.

1 Press function key 

.
Actual position screen (multi-path simultaneous display) appears.
2 Press soft key [(OPRT)].
3 Press continuous menu key  until soft key [SINGLE PATH] appears.



4 Press soft key [SINGLE PATH].
The single-path display appears as shown below:



5 Press soft key [MULTI PATHS]. The multi-path display appears.

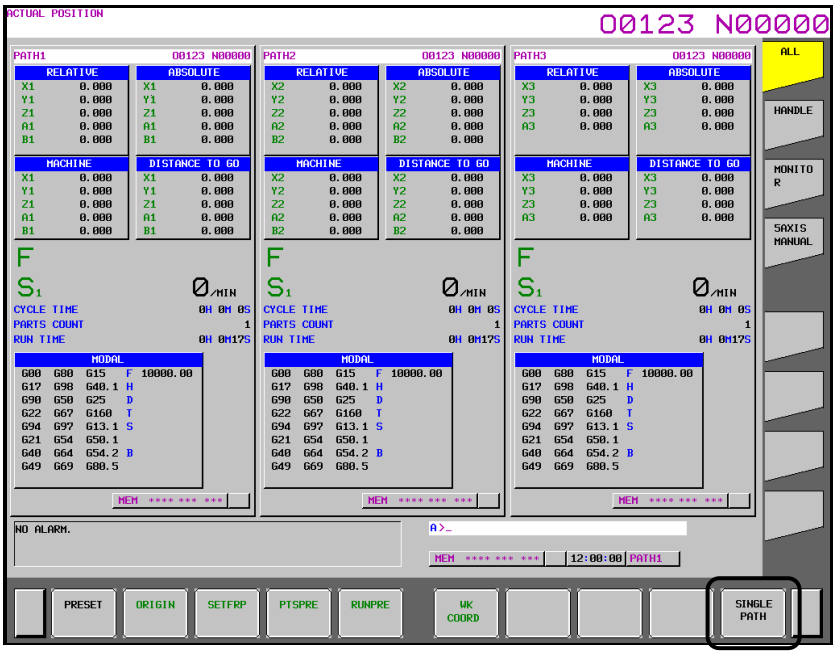
Procedure (15/19-inch display unit)

The procedure is explained using the current position display screen as an example.

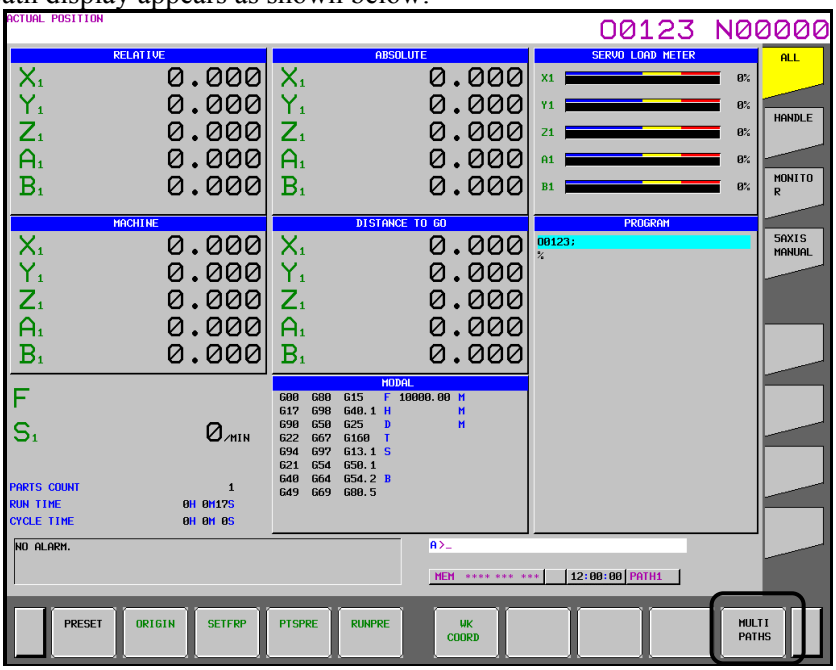
1 Press function key 

.

Actual position screen (multi-path simultaneous display) appears.



2 Press horizontal soft key [SINGLE PATH].
The single-path display appears as shown below:



3 Press horizontal soft key [MULTI PATHS]. The multi-path display appears.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11355					MTS			

[Input type] Parameter input
[Data type] Bit

#3 MTS The function for switching between simultaneous multi-path display and single-path display is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
11304								PGR

[Input type] Parameter input
[Data type] Bit

#0 PGR When the path select signal is changed, the screen of the multi path simultaneous display group:
0: Is not switched.
1: Is switched to the display group including the selected path.

13131	Group number for simultaneous display of multiple paths

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 10
This parameter sets a group for simultaneous display on one screen in a multi-path system.
The paths set to belong to the same group are displayed on one screen.
If the values for all paths are set to 0, the simultaneous multi-path display function is disabled.

NOTE
When specifying groups, specify group numbers not less than 1 successively.
On 8.4-inch and 10.4-inch display units, up to three paths can be specified for simultaneous display.
On a 15-inch and 19-inch display units, up to four paths can be specified for simultaneous display.

13132	Simultaneous multi-path display order number

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of paths included in a simultaneous multi-path display group
This parameter sets the display order of a path defined to belong to a simultaneous multi-path display group.
Set the order, using numbers ranging from 1 to the number of paths included in the simultaneous multi-path display group.

NOTE
Specify successive order numbers not less than 1 for the paths defined to belong to a group.

12.7 FIVE AXES DISPLAY IN ONE SCREEN FOR THE 8.4-INCH DISPLAY UNIT

The positions for up to five axes can be displayed simultaneously on one screen on a 8.4-inch display unit.
This function is enabled when bit 4 (9DE) of parameter No. 11350 is set to 1.
This function is enabled with the following screens:

- Absolute position display screen
- Relative position display screen
- Overall position display screen
- Handle interruption screen
- Operating monitor screen
- Program check screen
- Offset screen
- Macro screen
- Software operator's panel screen
- 4th/5th axis offset screen

Screen display

As shown in Fig. 12.7 (a), the coordinates for five axes are displayed on one screen on a 8.4-inch display unit.

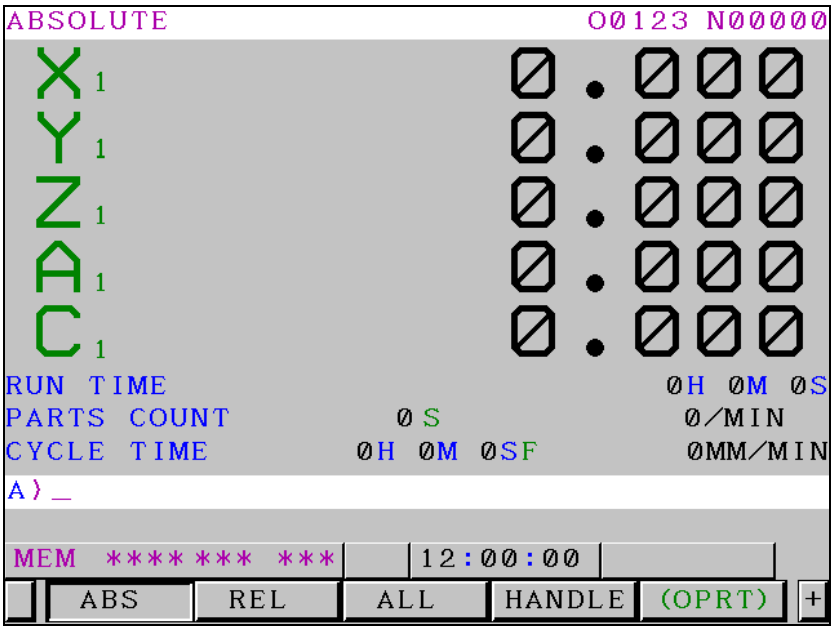


Fig. 12.7 (a) Absolute position display screen

When this function is enabled with the overall position display screen and handle interruption screen, the number of types of coordinates displayed on one screen decreases. The types of displayed coordinates can be changed by pressing MDI key 

and function key [ALL] or [HANDLE].

Fig. 12.7 (b) and Fig. 12.7 (c) show the displayed screens.

Overall position display screen

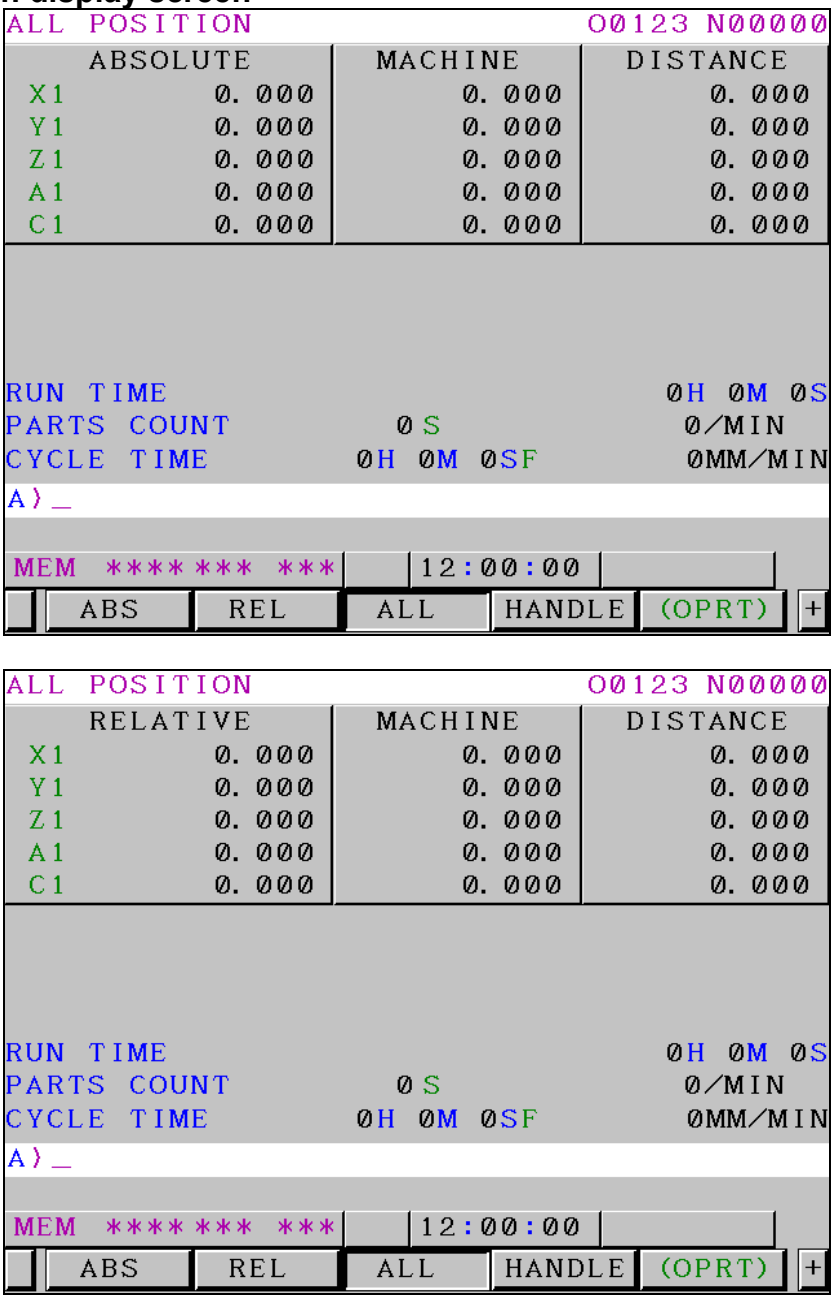
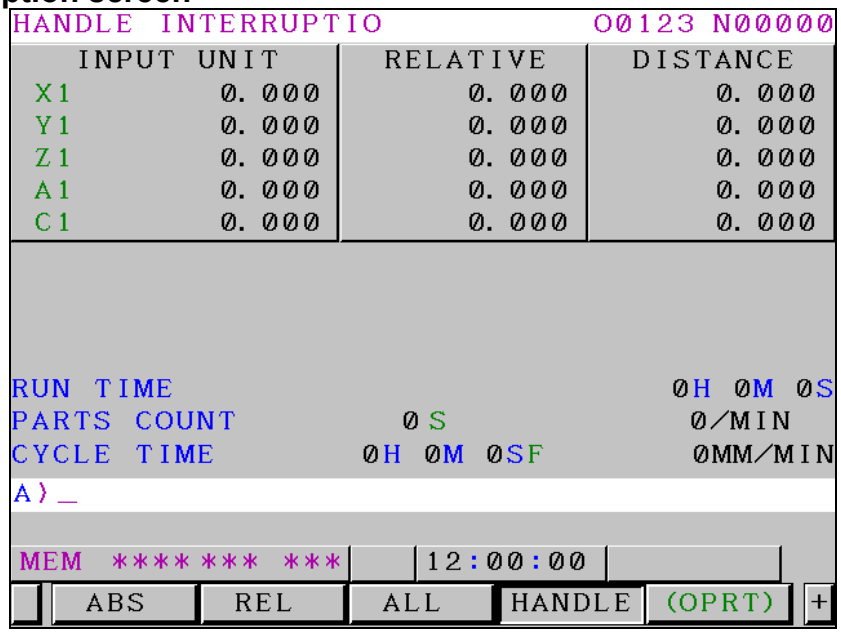


Fig. 12.7 (b) Overall position display screen

Handle interruption screen



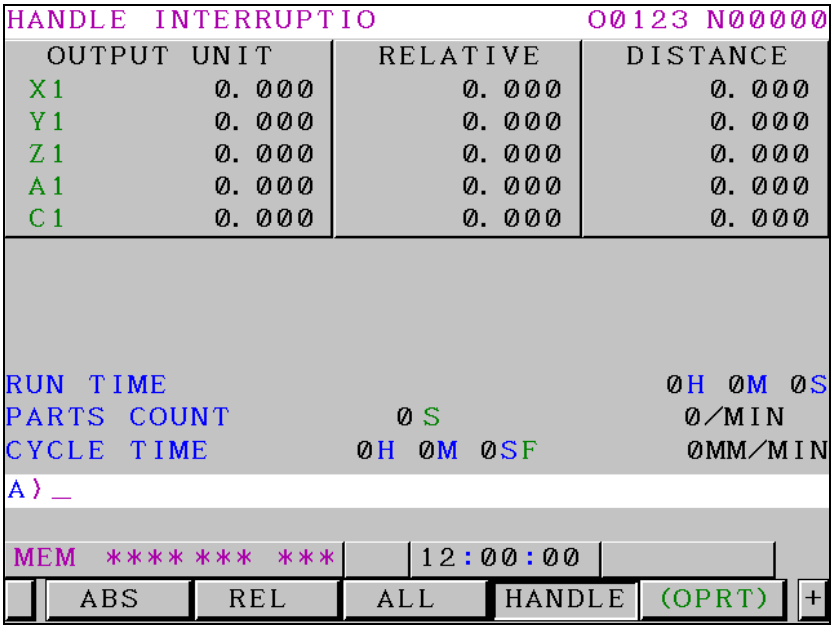


Fig. 12.7 (c) Handle interruption screen

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11350				9DE				

[Input type] Parameter input
[Data type] Bit

#4 9DE On 8.4-inch display unit, the maximum number of axes that can be displayed on a single screen is:
0: 4.
1: 5.

12.8 PATH NAME EXPANSION DISPLAY FUNCTION

The path name expansion display function enlarges the path name displayed at the upper right of the screen, which makes easy to check the currently selected path.

Explanation

When this function is enabled, the items on the screen are displayed as follows:

- The path name displayed in the status area is enlarged and displayed at the upper right of the screen. Only the alphanumeric characters in an arbitrary path name are displayed when enlarged (see (1) in Fig. 12.8 (a)). For the display colors, see the following paragraph.
- The O and N numbers are always displayed in small characters as shown in (2) in Fig. 12.8 (a).
  The displayed program names consist of the following number of characters:
  10.4-inch display unit: 17 characters (When a program number consists of more than 17 characters, the first 16 characters and a tilde (~) are displayed.)
  15-inch display unit: 32 characters (Maximum number of characters of an arbitrary program name)
- The path name displayed in the status area is displayed as usual (see (3) in Fig. 12.8 (a)).

This function is enabled by setting bit 2 (PNE) of parameter No. 11350.
With all screens other than those for the C Language Executor and macro executor (conversational macro), this function is available.

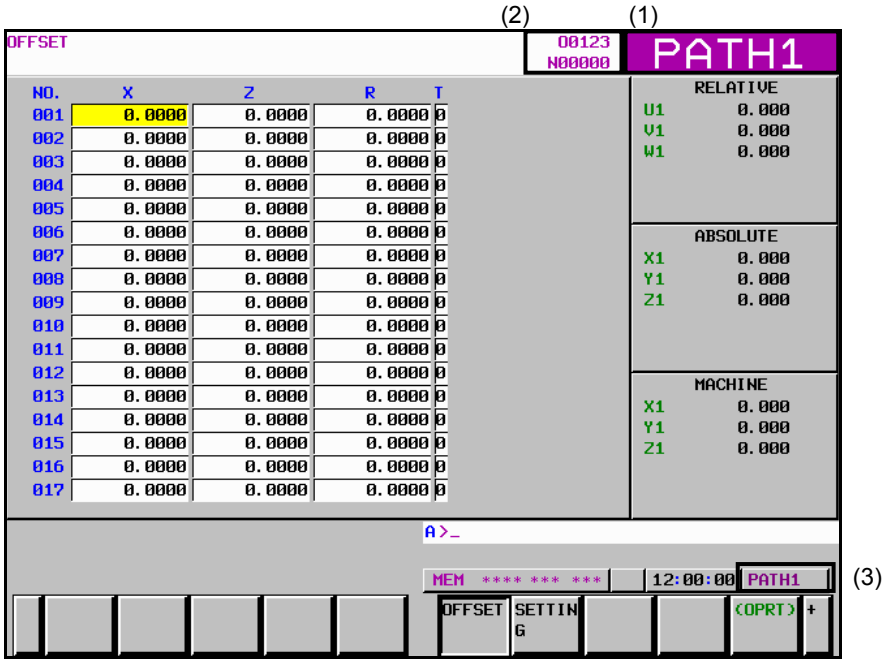


Fig. 12.8 (a) Sample screen when the path name expansion display function is enabled (offset screen)

Display character color

The following colors specified on the color setting screen are used for the enlarged display characters:

Character color: No. 3 color (title color)
Background color: No. 10 color (title background color)

The path name can be displayed in reverse video by setting bit 0 (PNI) of parameter No. 11352.

Table 12.8 (a) Enlarged display character colors and parameters

		Character color	Background color
Bit 0 (PNI) of parameter No. 11352	0: A normal display	No. 3 color	No. 10 color
	1: A reverse display	No. 10 color	No. 3 color

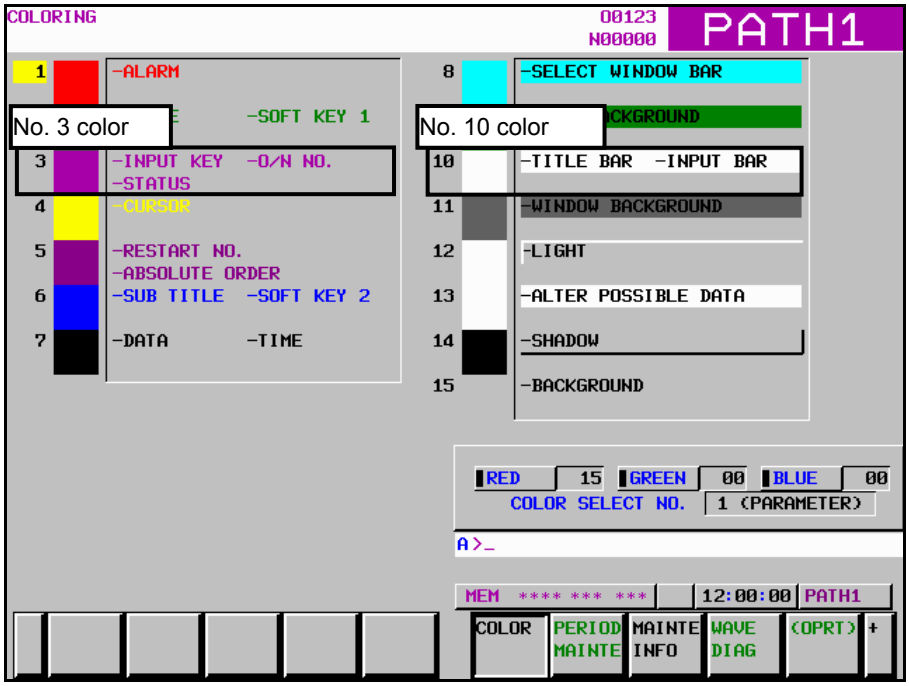


Fig. 12.8 (b) Character color numbers for the path name expansion display function

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11350						PNE		

[Input type] Parameter input
[Data type] Bit

#2 PNE Path name expansion display function is:
0: Disabled.
1: Enabled.

NOTE
1 When this parameter is set, the power must be turned off before operation is continued.
2 This parameter is invalid if the number of paths is 1.
3 It is effective to 10.4/15/19-inch display units.

	#7	#6	#5	#4	#3	#2	#1	#0
11352								PNI

[Input type] Parameter input
[Data type] Bit path

#0 PNI The display by the path name enlarged display function is:
0: A normal display.
1: A reverse display.

NOTE
This parameter is effective to 10.4/15/19-inch display units.

3141	Path name (1st character)
3142	Path name (2nd character)
3143	Path name (3rd character)
3144	Path name (4th character)
3145	Path name (5th character)
3146	Path name (6th character)
3147	Path name (7th character)

[Input type] Parameter input
[Data type] Word path
[Valid data range] See the character-code correspondence table.
Specify a path name with codes.
Any character string consisting of alphanumeric characters, katakana characters, and special characters with a maximum length of seven characters can be displayed as a series name.

NOTE
1 For characters and codes, see Appendix, “CHARACTER CODE LIST”.
2 When 0 is set in parameter No. 3141, PATH1(,PATH2...) are displayed as path names.
3 When optional path name display is enlarged (with bit 2 (PNE) of parameter No. 11350 set to 1), only alphanumeric characters are displayed. If any other type of characters are set, spaces are displayed instead.

12.9 SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION

Overview

Keeping the same characters displayed in the same positions on the screen for a long time will shorten the life of the LCD.
To prevent this, the CNC screen can be erased. The screen erasure function allows the user to perform a key operation to erase the screen. The automatic screen erasure function erases the screen automatically when there has been no key operation for a parameter-set period of time.

Screen erasure function

When 0 is set in parameter No. 3123, the CNC screen can be erased by pressing the 

key and any function key (such as 

or 

) at the same time. The CNC screen can be displayed again by pressing any function key.

Automatic screen erasure function

When there has been no key operation for a time (in minutes) set in parameter No. 3123, the CNC screen is erased automatically. The CNC screen is displayed again by pressing a key.

- Screen erasure by the automatic screen erasure function

If the following conditions are all satisfied for the time (in minutes) set in parameter No. 3123, the CNC screen is erased.

Conditions for automatically erasing the CNC screen

- Parameter No. 3123 ≠ 0
- None of the following key operations is performed.
  MDI keys
  Soft keys
  External key input
- No alarm is issued.

- Redisplay of the screen by the automatic screen erasure function

If one of the following conditions is satisfied when the CNC screen is off, the CNC screen is displayed again:

Conditions for redisplaying the CNC screen

- One of the following key operations is performed.
  MDI keys
  Soft keys
  External key input
- An alarm is issued.

- Screen erasure by using the  key + function key

When a non-zero value is set in parameter No. 3123, the screen is not erased with the  key and a function key.

- Set time

Only the time set in parameter No. 3123 for path 1 is valid.

- Alarm in another path

When an alarm is issued in any of the paths, the screen is not erased.

Parameter

3123	Time required before a screen saver is activated

[Input type] Setting input
[Data type] Byte path
[Unit of data] min
[Valid data range] 0 to 127

After a time (in minutes) set in parameter No. 3123 passes without key operation, the NC screen is erased automatically. Pressing a key causes the NC screen to reappear.

NOTE
1 Setting 0 disables automatic screen erasure.
2 This function cannot be used together with manual screen erasure. If 1 or a larger value is set in this parameter, manual screen erasure is disabled.

12.10 LOAD METER SCREEN

Overview

The servo and spindle load meters can be displayed in place of the modal code display part and the remaining travel distance part of the current position display on the program check screen.
This function is enabled only when the 10.4-inch display unit is used.

12.10.1 Single-path Display

Screen layout

In the single-path display mode, the servo or spindle load meter is displayed in place of the remaining travel distance display of the total position display.

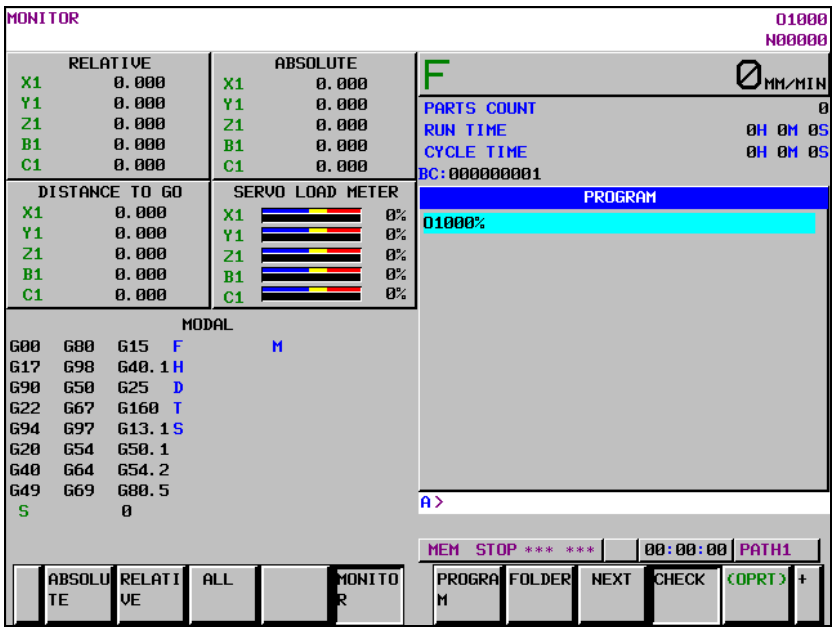


Fig. 12.10.1 (a) Servo load meter

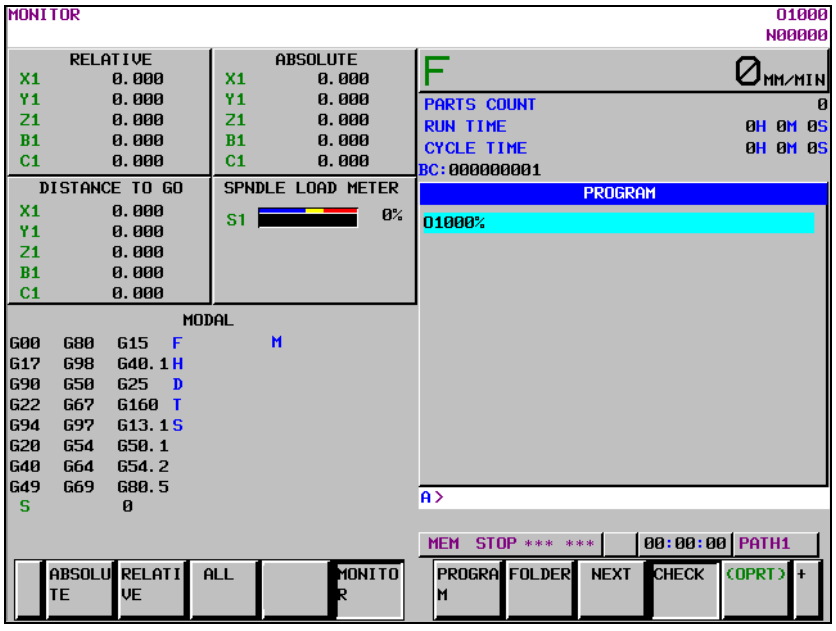
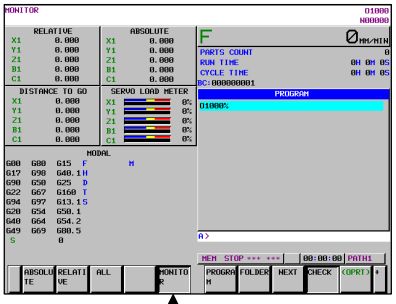


Fig. 12.10.1 (b) Spindle load meter

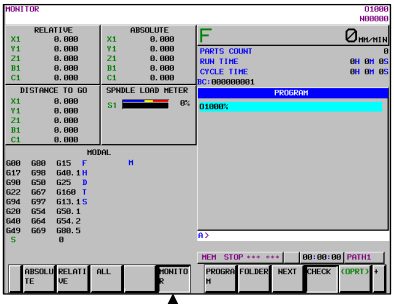
Screen switching

The servo load meter and spindle load meter are displayed by pressing soft key [MONITOR] on the left side of the screen. Initially the servo load meter is displayed. Each time soft key [MONITOR] is pressed, the servo load meter and spindle load meter alternate.

Servo load meter



Spindle load meter



Switched by pressing soft key MONITOR]

12.10.2 Two-path Display and Three-path Display

Screen layout

In the two- or three-path display mode, the modal information display part on the program check screen can be replaced by the servo or spindle load meter.

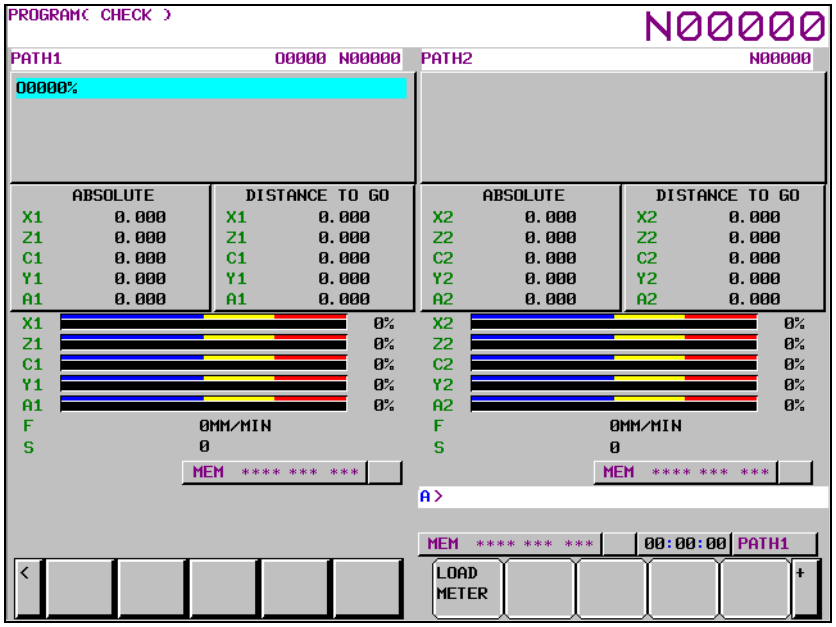


Fig. 12.10.2 (a) Servo load meter screen

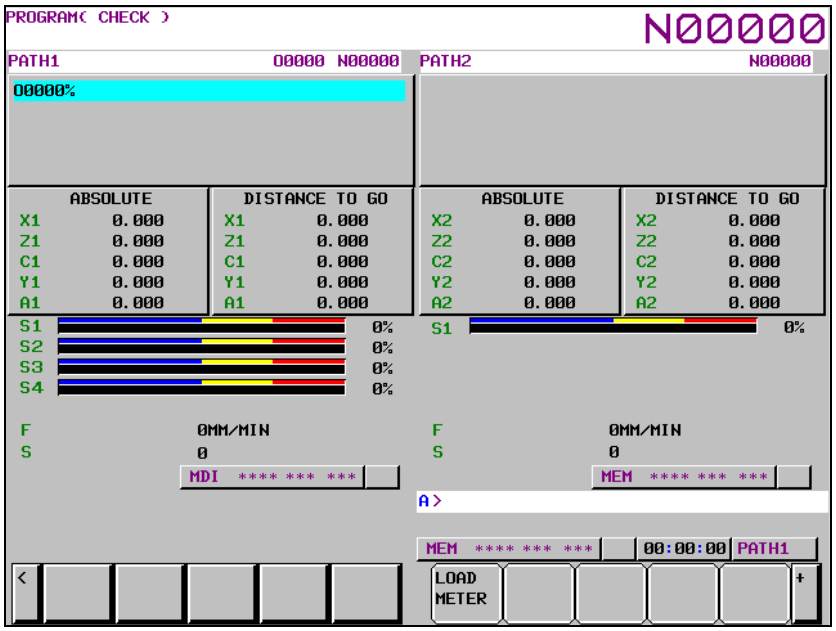
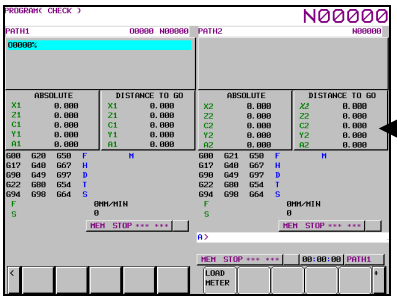


Fig. 12.10.2 (b) Spindle load meter screen

Screen switching

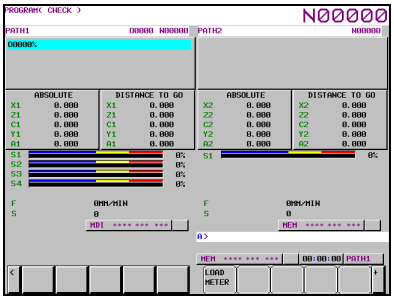
By pressing soft key [LOAD METER], the screen display can be switched among the modal information display, servo load meter display, and spindle load meter display.

Modal information



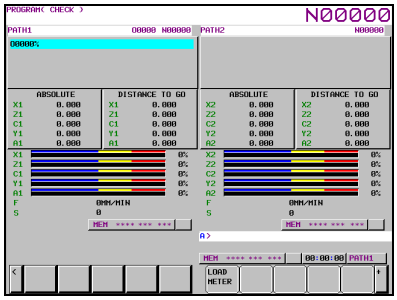
Press soft key [LOAD METER].

Spindle load meter



Press soft key [LOAD METER].

Servo load meter



Press soft key [LOAD METER].

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3192	PLD							

[Input type] Parameter input
[Data type] Bit

#7 PLD When the current position is indicated for a path, and when the program check screen is displayed in a two- or three-path system, the function for displaying servo load meters and spindle load meters is :

0: Disabled.
1: Enabled.

13140	First character in spindle load meter display

13141	Second character in spindle load meter display

[Input type] Setting input
[Data type] Byte spindle
[Valid data range] These parameters set character codes to set the name of each spindle that appears in spindle load meter display. Any character string consisting of numeric characters, alphabetical characters, katakana characters, and symbols with a maximum length of two characters can be displayed as a spindle name.
If 0 is set, the following is displayed:

1st spindle S1
2nd spindle S2
3rd spindle S3
4th spindle S4

12.11 DISPLAYING THE PROGRAM NUMBER/NAME, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION

The program number, program name, sequence number, and current CNC status are always displayed on the screen except when the power is turned on, a system alarm occurs.
If data setting or the input/output operation is incorrect, the CNC does not accept the operation and displays a warning message.
This section describes the display of the program number, sequence number, and status, and warning messages displayed for incorrect data setting or input/output operation.

Screens of a 10.4-inch display unit

12.11.1 Displaying the Program Number, Program Name, and Sequence Number

The number and name of the program currently selected or currently executed and the current sequence number are indicated in the upper right part as shown below.

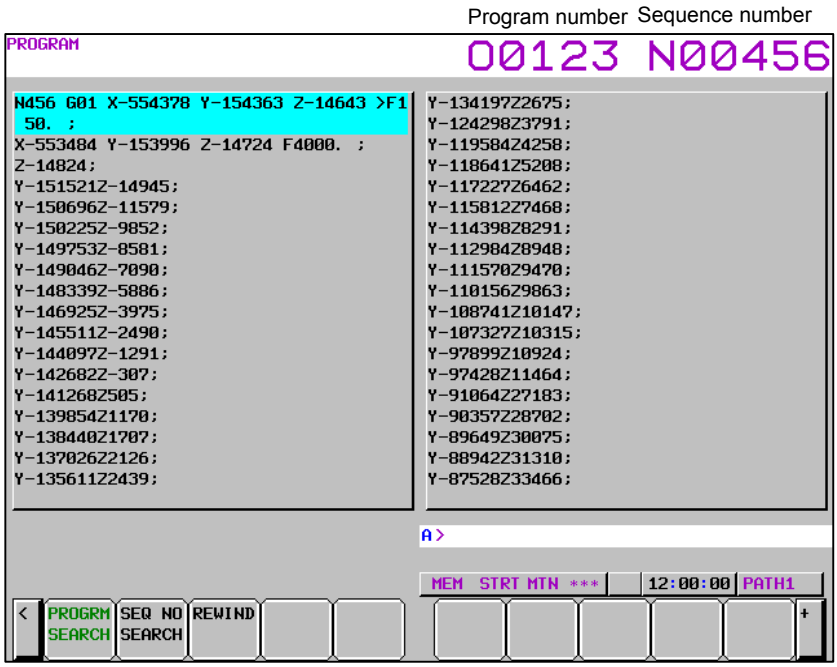


Fig. 12.11.1 (a) Program number and sequence number (10.4-inch display unit)

In the EDIT mode, the number and name of the program currently edited in the foreground are indicated.

When 10.4-inch display unit is used, the program number that is displayed in the upper right of the screen is changed according to the digit of the program number.
When the program number is 4-digit or less, “O” plus a 4-digit numeric is displayed.

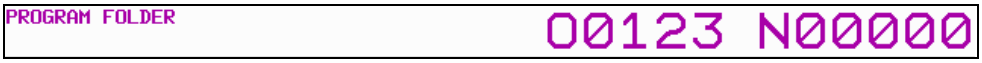


Fig. 12.11.1 (b) Program number is 4-digit or less

When the program number is 5-digit or more, “O” plus a 8-digit numeric is displayed.

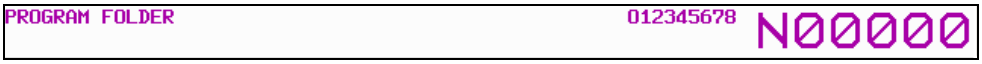


Fig. 12.11.1 (c) Program number is 5-digit or more

12.11.2 Displaying the Status and Warning for Data Setting or Input/Output Operation

The current mode, automatic operation state, alarm state, and program editing state are displayed on the next to last line on the screen allowing the operator to readily understand the operation condition of the system. If data setting or the input/output operation is incorrect, the CNC does not accept the operation and a warning message is displayed on the next to last line of the screen. This prevents invalid data setting and input/output errors.

Explanation

- **Description of each display**

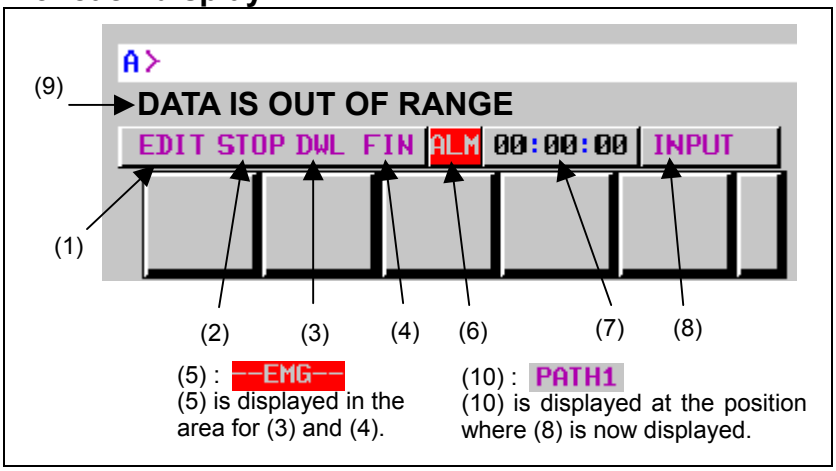


Fig. 12.11.2 (a) Positions of status indications

(1) Current mode

MDI : Manual data input, MDI operation
MEM : Automatic operation (memory operation)
RMT : Automatic operation (DNC operation, or such like)
EDIT : Memory editing
HND : Manual handle feed
JOG : Jog feed
INC : Manual incremental feed
REF : Manual reference position return
**** : Indicates a mode other than the above.

(2) Automatic operation status

**** : Reset (When the power is turned on or the state in which program execution has terminated and automatic operation has terminated.)
STOP : Automatic operation stop (The state in which one block has been executed and automatic operation is stopped.)
HOLD : Feed hold (The state in which execution of one block has been interrupted and automatic operation is stopped.)
STRT : Automatic operation start-up (The state in which the system operates automatically)
MSTR : Manual numerical command start state (The state in which a manual numerical command is being executed)

Alternatively, tool retract and recover operation state (The state in which a recover operation and repositioning operation are being performed)

(3) Axis moving status/dwell status

MTN : Indicates that the axis is moving.
DWL : Indicates the dwell state.
*** : Indicates a state other than the above.

(4) State in which an auxiliary function is being executed

FIN : Indicates the state in which an auxiliary function is being executed. (Waiting for the complete signal from the PMC)

(5) Emergency stop or reset status

--EMG-- : Indicates emergency stop.(Blinks in reversed display.)
-RESET- : Indicates that the reset signal is being received.

(6) Alarm status

ALM : Indicates that an alarm is issued. (Blinks in reversed display.)
BAT : Indicates that the voltage of the lithium battery (the backup battery of the CNC) has decreased. (Blinks in reversed display.)
APC : Indicates that the voltage of the backup battery of the absolute pulse coder has decreased. (Blinks in reversed display.)
FAN : Indicates that the rotation speed of the fan has decreased. (Blinks in reversed display.)
LKG : Indicates that the insulation resistance value of the motor or the power line has decreased.. (Blinks in reversed display.)
PSW : Indicates that the power supply is abnormal state. (Blinks in reversed display.)
PMC : Indicates that a PMC alarm is issued. (Blinks in reversed display.)
Space : Indicates a state other than the above.

(7) Current time

hh : mm : ss - Hours, minutes, and seconds

(8) Program editing status

INPUT : Indicates that data is being input.
OUTPUT : Indicates that data is being output.
SEARCH : Indicates that a search is being performed.
EDIT : Indicates that another editing operation is being performed (insertion, modification, etc.)
LSK : Indicates that labels are skipped when data is input.
RSTR : Indicates that the program is being restarted
COMPARE : Indicates that a data comparison is being made.
OFST : Indicates that the tool length compensation amount measurement mode is set (for the machining center system) or that the tool length compensation amount write mode is set (for the lathe system).
WOFS : Indicates that the workpiece origin offset amount measurement mode is set.
AICC1 : Indicates that operation is being performed in the AI contour control I mode.
AICC2 : Indicates that operation is being performed in the AI contour control II mode.
MEM-CHK : Indicates that a program memory check is being made.
WSFT : Indicates that the workpiece shift amount write mode is set.
LEN : Indicates that the active offset value change mode (tool length offset value of the M series) is set.
RAD : Indicates that the active offset value change mode (tool radius compensation amount of the M series) is set.

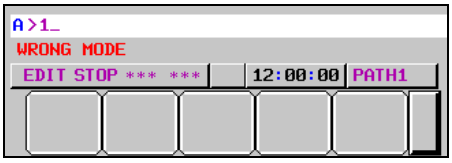
WZR	:	Indicates that the active offset value change mode (workpiece origin offset value) is set.
TOFS	:	Indicates that the active offset value change mode (tool offset value of the T series) is set.
OFSX	:	Indicates that the active offset value change mode (X-axis tool offset value of the T series) is set.
OFSZ	:	Indicates that the active offset value change mode (Z-axis tool offset value of the T series) is set.
OFSY	:	Indicates that the active offset value change mode (Y-axis tool offset value of the T series).
TCP	:	Indicates that operation is being performed in the tool center point control.
TWP	:	Indicates that operation is being performed in the tilted working plane command mode.
Space	:	Indicates that no editing operation is being performed.

(9) Warning for data setting or input/output operation

When invalid data is entered (wrong format, value out of range, etc.), when input is disabled (wrong mode, write disabled, etc.), or when input/output operation is incorrect (wrong mode, etc.), a warning message is displayed. When the RS232-C communication port is being used, “CANNOT USE I/O DEVICE” is displayed. In this case, the CNC does not accept the setting or input/output operation (retry the operation according to the message).

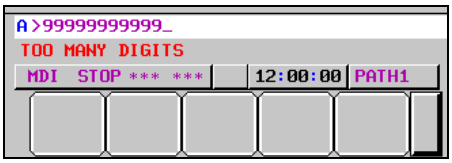
Example 1)

When a parameter is entered



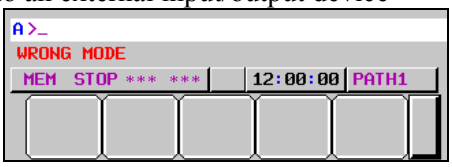
Example 2)

When a parameter is entered



Example 3)

When a parameter is output to an external input/output device



(10) Tool post name

The number of a path whose status is indicated is displayed.

PATH1 : Indicates that the status being indicated is for path 1.
Other names can be used depending on the settings of parameters Nos. 3141 to 3147. The tool post name is displayed at the position where (8) is now displayed. While the program is edited, (8) is displayed.

Screens of a 15/19-inch display unit

12.11.3 Displaying the Program Number, Program Name, and Sequence Number (15/19-inch Display Unit)

The number and name of the program currently selected or currently executed and the current sequence number are indicated in the upper right part as shown below.

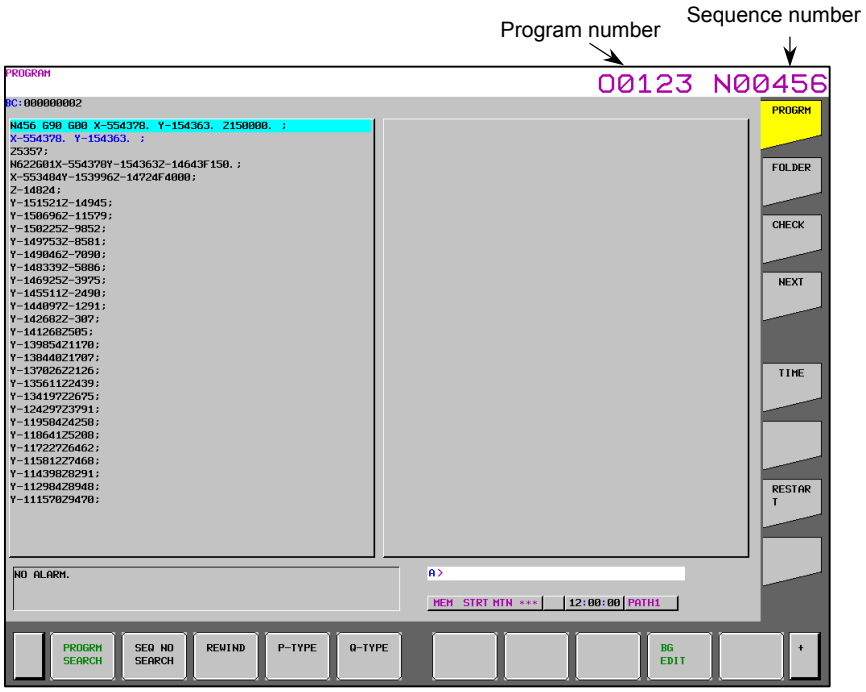


Fig. 12.11.3 (a) Program number and sequence number (15-inch display unit)

In the EDIT mode, the number and name of the program currently edited in the foreground are indicated.

12.11.4 Displaying the Program Name

When bit 4 (DPC) of parameter No. 11354 is set to 0, the program name (comment) of main program is displayed between the screen title and O number.

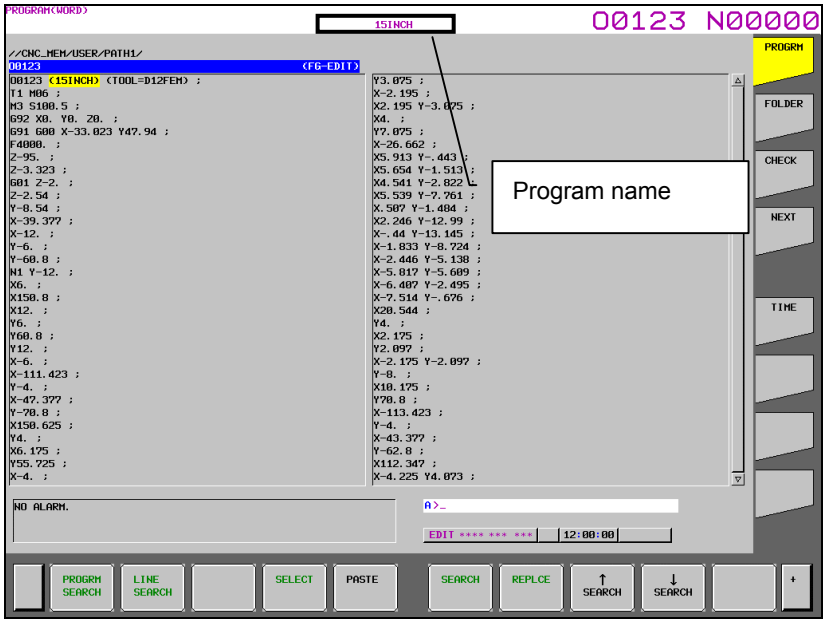


Fig. 12.11.4 (a) Program name (15inch)

A program name consisting of up to 18 characters can be displayed.
If a program name is longer than 18 characters, the first 17 characters are displayed. A tilde (~) is displayed as the 18th character, which indicates that the program name is abbreviated.

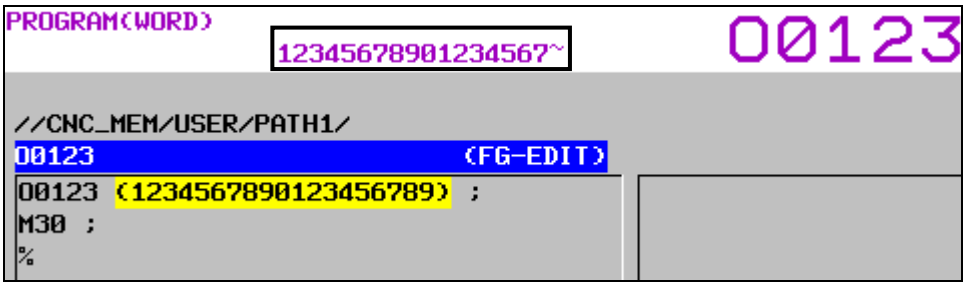


Fig. 12.11.4 (b) Program name longer than 18 characters

12.11.5 Displaying the Status and Warning for Data Setting or Input/Output Operation (15/19-inch Display Unit)

The current mode, automatic operation state, alarm state, and program editing state are displayed on the next to last line on the screen allowing the operator to readily understand the operation condition of the system. If data setting or the input/output operation is incorrect, the CNC does not accept the operation and a warning message is displayed on the next to last line of the screen. This prevents invalid data setting and input/output errors.

Explanation

- Description of each display

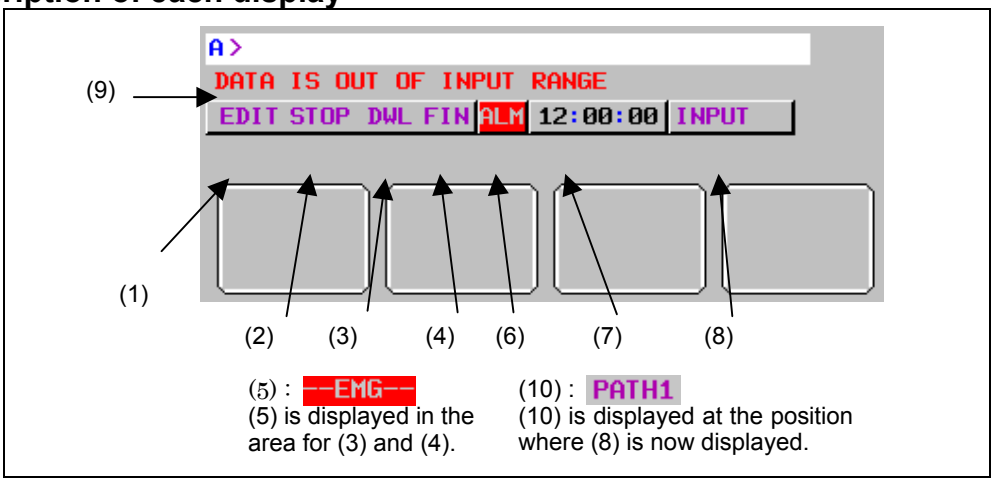


Fig. 12.11.5 (a) Positions of status indications

(1) Current mode

MDI : Manual data input, MDI operation
MEM : Automatic operation (memory operation)
RMT : Automatic operation (DNC operation, or such like)
EDIT : Memory editing
HND : Manual handle feed
JOG : Jog feed
INC : Manual incremental feed
REF : Manual reference position return
**** : Indicates a mode other than the above.

(2) Automatic operation status

**** : Reset (When the power is turned on or the state in which program execution has terminated and automatic operation has terminated.)
STOP : Automatic operation stop (The state in which one block has been executed and automatic operation is stopped.)
HOLD : Feed hold (The state in which execution of one block has been interrupted and automatic operation is stopped.)
STRT : Automatic operation start-up (The state in which the system operates automatically)
MSTR : Manual numerical command start state (The state in which a manual numerical command is being executed)
Alternatively, tool retract and recover operation state (The state in which a recover operation and repositioning operation are being performed)

(3) Axis moving status/dwell status

MTN : Indicates that the axis is moving.
DWL : Indicates the dwell state.

*** : Indicates a state other than the above.

(4) State in which an auxiliary function is being executed

FIN : Indicates the state in which an auxiliary function is being executed. (Waiting for the complete signal from the PMC)
*** : Indicates a state other than the above.

(5) Emergency stop or reset status

--EMG-- : Indicates emergency stop.(Blinks in reversed display.)
-RESET- : Indicates that the reset signal is being received.

(6) Alarm status

ALM : Indicates that an alarm is issued. (Blinks in reversed display.)
BAT : Indicates that the voltage of the lithium battery (the backup battery of the CNC) has decreased. (Blinks in reversed display.)
APC : Indicates that the voltage of the backup battery of the absolute pulse coder has decreased. (Blinks in reversed display.)
FAN : Indicates that the rotation speed of the fan has decreased. (Blinks in reversed display.)
LKG : Indicates that the insulation resistance value of the motor or the power line has decreased.. (Blinks in reversed display.)
PSW : Indicates that the power supply is abnormal state. (Blinks in reversed display.)
PMC : Indicates that a PMC alarm is issued. (Blinks in reversed display.)
Space : Indicates a state other than the above.

(7) Current time

hh : mm : ss - Hours, minutes, and seconds

(8) Program editing status

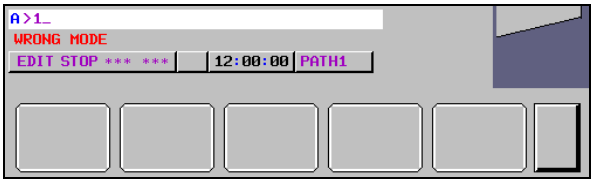
INPUT : Indicates that data is being input.
OUTPUT : Indicates that data is being output.
SEARCH : Indicates that a search is being performed.
EDIT : Indicates that another editing operation is being performed (insertion, modification, etc.)
LSK : Indicates that labels are skipped when data is input.
RSTR : Indicates that the program is being restarted
COMPARE : Indicates that a data comparison is being made.
OFST : Indicates that the tool length compensation amount measurement mode is set (for the machining center system) or that the tool length compensation amount write mode is set (for the lathe system).
WOFS : Indicates that the workpiece origin offset amount measurement mode is set.
AICC1 : Indicates that operation is being performed in the AI contour control I mode.
AICC2 : Indicates that operation is being performed in the AI contour control II mode.
MEM-CHK : Indicates that a program memory check is being made.
WSFT : Indicates that the workpiece shift amount write mode is set.
LEN : Indicates that the active offset value change mode (tool length offset value of the M series) is set.
RAD : Indicates that the active offset value change mode (tool radius compensation amount of the M series) is set.
WZR : Indicates that the active offset value change mode (workpiece origin offset value) is set.
TOFS : Indicates that the active offset value change mode (tool offset value of the T series) is set.
OFSX : Indicates that the active offset value change mode (X-axis tool offset value of the T series) is set.

OFSZ : Indicates that the active offset value change mode (Z-axis tool offset value of the T series) is set.
OFSY : Indicates that the active offset value change mode (Y-axis tool offset value of the T series).
TCP : Indicates that operation is being performed in the tool center point control.
TWP : Indicates that operation is being performed in the tilted working plane command mode.
Space : Indicates that no editing operation is being performed.

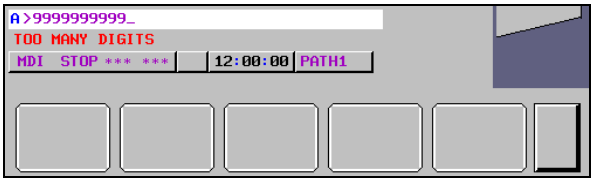
(9) Warning for data setting or input/output operation

When invalid data is entered (wrong format, value out of range, etc.), when input is disabled (wrong mode, write disabled, etc.), or when input/output operation is incorrect (wrong mode, etc.), a warning message is displayed. When the RS232-C communication port is being used, “CANNOT USE I/O DEVICE” is displayed. In this case, the CNC does not accept the setting or input/output operation (retry the operation according to the message).

Example 1)
When a parameter is entered



Example 2)
When a parameter is entered



Example 3)
When a parameter is output to an external input/output device



(10) Tool post name

The number of a path whose status is indicated is displayed.
PATH1 : Indicates that the status being indicated is for path 1.
Other names can be used depending on the settings of parameters Nos. 3141 to 3147. The tool post name is displayed at the position where (8) is now displayed. While the program is edited, (8) is displayed.

13 GRAPHIC FUNCTION

Chapter 13, "GRAPHIC FUNCTION", consists of the following sections:

13.1 GRAPHIC DISPLAY 1743
13.2 DYNAMIC GRAPHIC DISPLAY 1754

13.1 GRAPHIC DISPLAY

The graphic display functions enable drawing of the tool path of the program currently used for machining.
These functions are intended to display the movement of the tool during automatic operation or during manual operation.
This enables the operator to check the progress of machining and the current tool position.
These functions include the following:

- The current tool position in the workpiece coordinate system is displayed.
- Graphic coordinates can be set freely.
- Rapid traverse and cutting feed can be drawn using a different color for each.
- The values of F, S, and T in the program during drawing are displayed.
- Graphic enlargement or reduction is possible.

Graphic display procedure

Explanation

Press the function key 

then press the [GRAPH] soft key to display the tool path graphic screen.

- Tool path graphic screen

The tool path graphic screen consists of three major areas:

- Graphic area for drawing a tool path
- Area for displaying machining information such as tool position information
- Area for displaying a graphic coordinate system

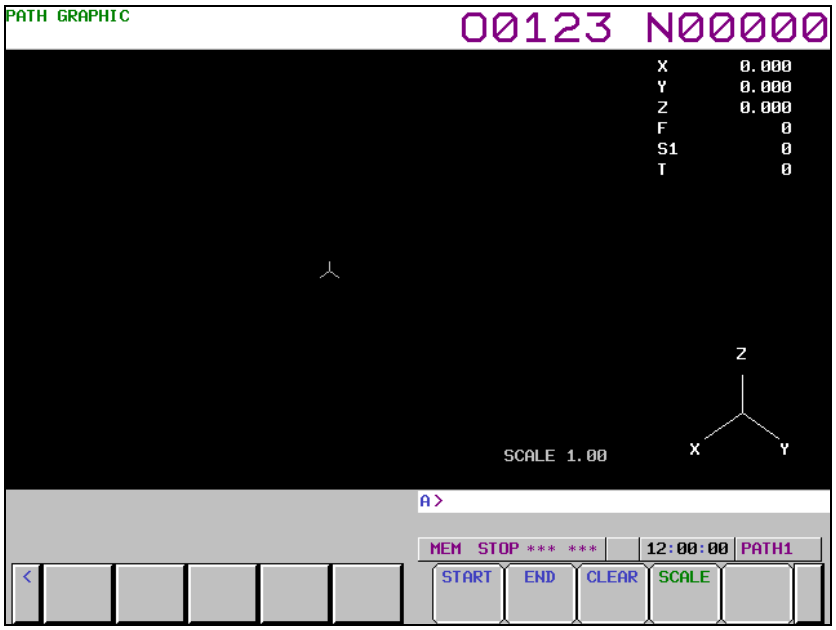


Fig. 13.1 (a) Tool path graphic screen (M series)

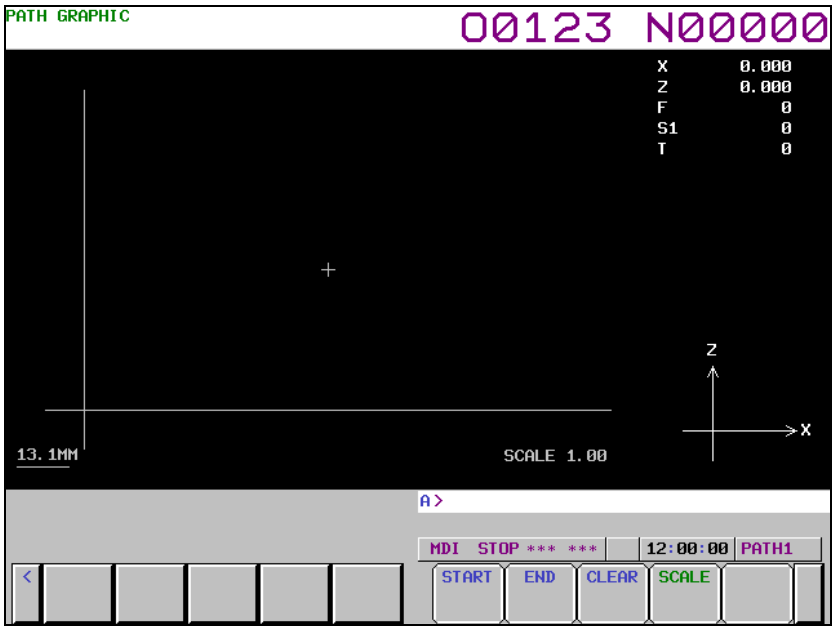


Fig. 13.1 (b) Tool path graphic screen (T series)

- Tool path

In a graphic coordinate system set by the graphic parameters described later, a tool path in the workpiece coordinate system is drawn.
Even when the tool position changes discontinuously for a cause such as the setting of the origin and the switching of the workpiece coordinate system, drawing is performed assuming that the tool has moved.
A tool path is continuously drawn even when the screen display is changed to another screen.

- Machining information

On the right side of the screen, the positions (along only three axes used for drawing) in the workpiece coordinate system, feedrate (F), spindle speed (S), and tool number (T) are displayed.

NOTE
Up to three graphic axes are used with the M series, and up to two graphic axes are used with the T series.

- **Graphic coordinate system**

On the lower-right portion of the screen, the coordinate axes and axis names of the graphic coordinate system are displayed.

Graphic parameter screen

Explanation

Press the function key 

then press the [PARAM] soft key to display the tool path graphic screen.

On the graphic parameter screen, make settings necessary for drawing a tool path.
The graphic parameter screen consists of three pages.

M

- **Graphic parameter screen page 1**

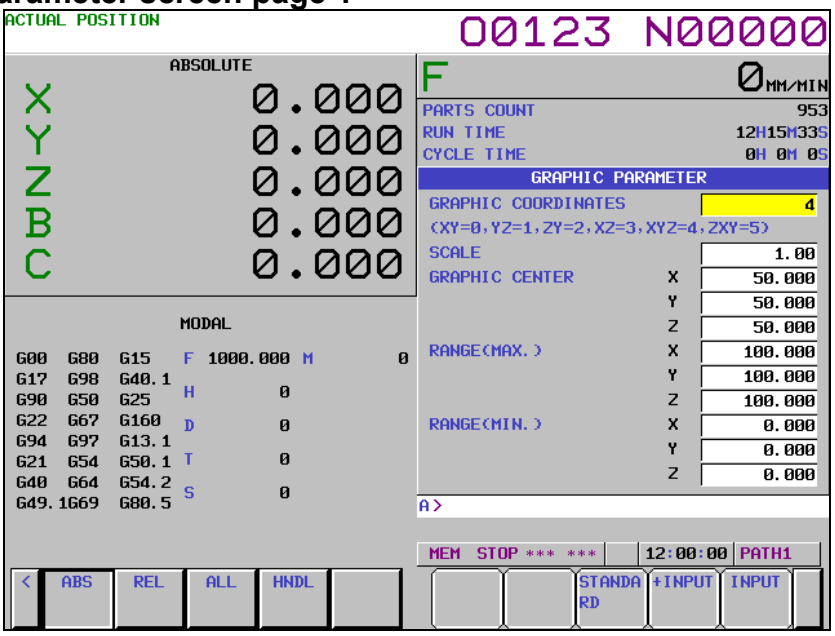


Fig. 13.1 (c) Graphic parameter screen page 1

On graphic parameter screen page 1, a graphic coordinate system, graphic range, and so forth are set.
In the setting of a graphic coordinate system, the coordinate axes and axis names of the set coordinate system are graphically displayed. When a 3-dimensional coordinate system is displayed, a rotation angle is also displayed.
A graphic range is set using one of two methods. One method sets a graphic scale and graphic center coordinates. The other sets the maximum values and minimum values of a graphic range.

- **Graphic parameter screen page 2**

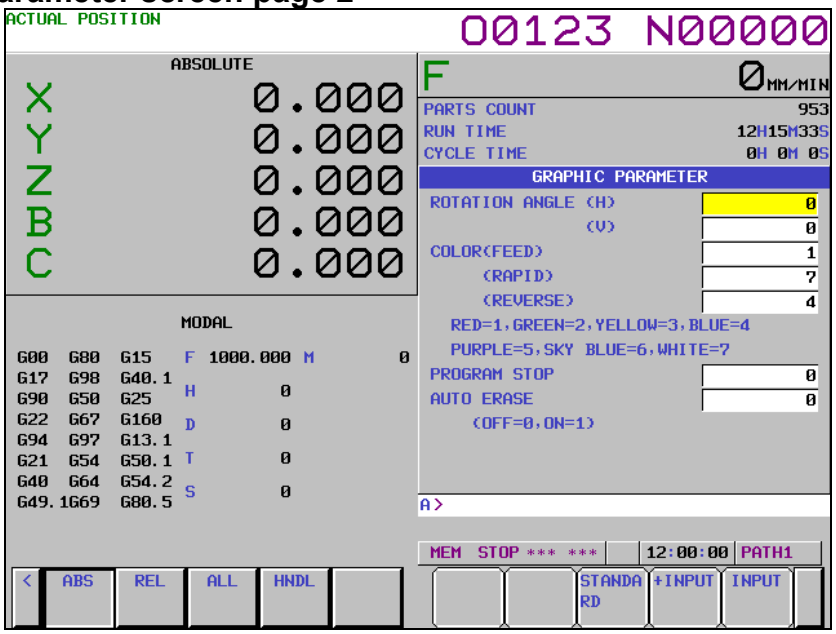


Fig. 13.1 (d) Graphic parameter screen page 2

On graphic parameter screen page 2, graphic colors, rotation angles, and whether to perform automatic erase operation are set.

- **Graphic parameter screen page 3**

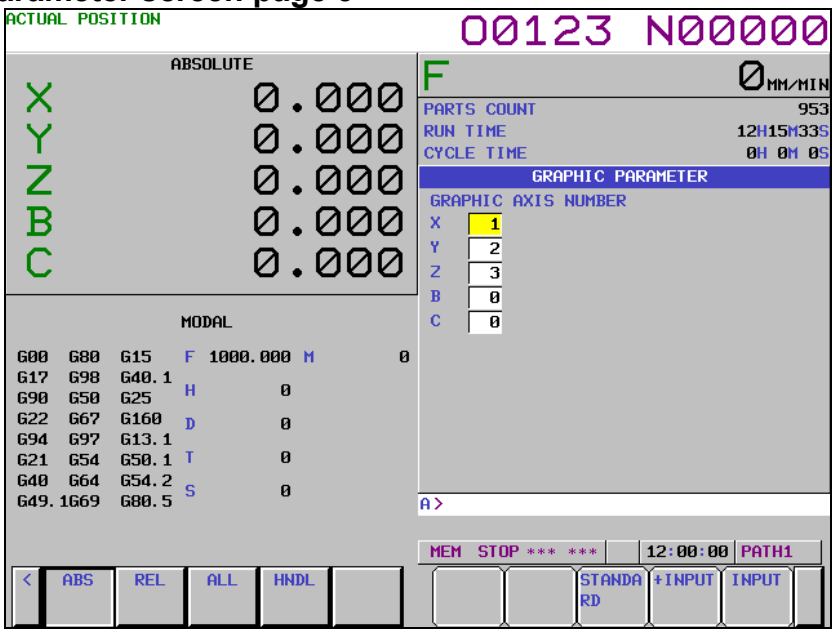


Fig. 13.1 (e) Graphic parameter screen page 3

On graphic parameter screen page 3, coordinate axes to be used for drawing are set.

T

- **Graphic parameter screen page 1**

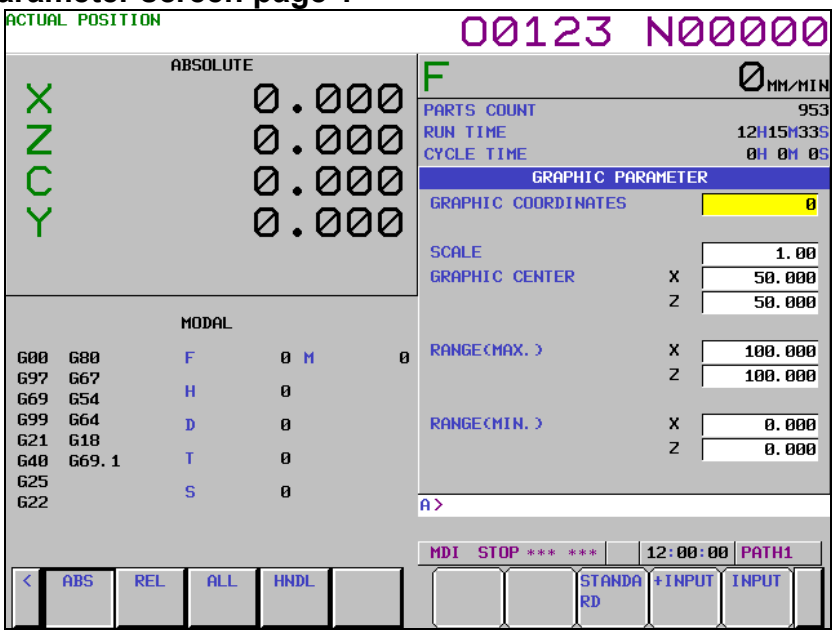


Fig. 13.1 (f) Graphic parameter screen page 1

On graphic parameter screen page 1, a graphic coordinate system, graphic range, and so forth are set.
In the setting of a graphic coordinate system, the coordinate axes and axis names of the set coordinate system are graphically displayed. When a 3-dimensional coordinate system is displayed, a rotation angle is also displayed.
A graphic range is set using one of two methods. One method sets a graphic scale and graphic center coordinates. The other sets the maximum values and minimum values of a graphic range.

- **Graphic parameter screen page 2**

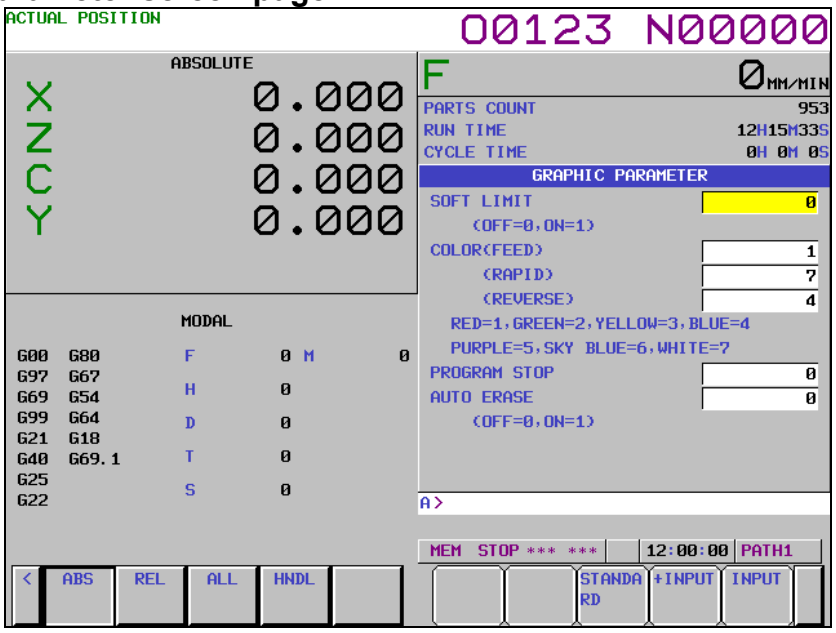


Fig. 13.1 (g) Graphic parameter screen page 2 (T series)

On graphic parameter screen page 2, graphic colors and whether to perform automatic erase operation are set.

- Graphic parameter screen page 3

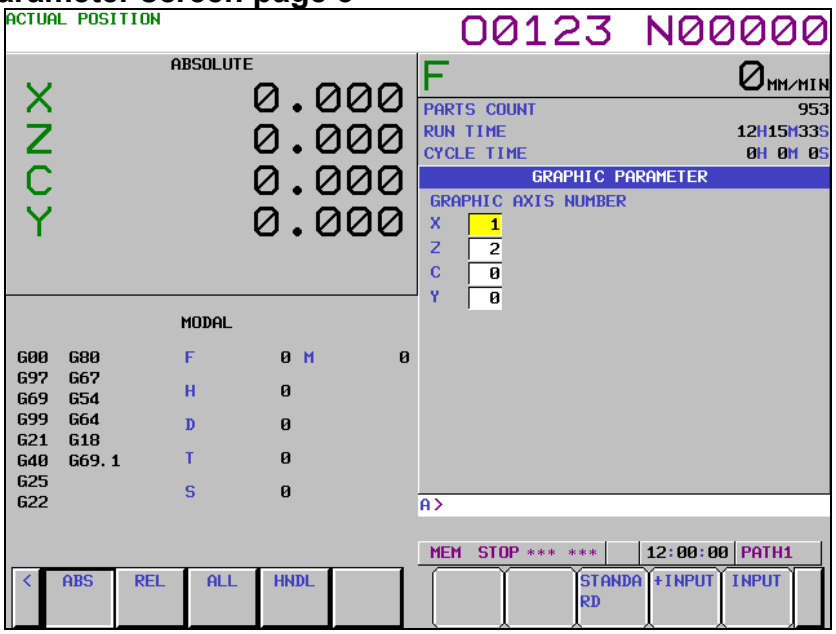


Fig. 13.1 (h) Graphic parameter screen page 3

On graphic parameter screen page 3, coordinate axes to be used for drawing are set.

Graphic parameter setting

Explanation

For tool path drawing, a graphic coordinate system, tool path graphic colors, and graphic range need to be set on the graphic parameter screen.
The graphic parameters to be set on the graphic parameter screen are described below.
When a value is set for a parameter, the parameter value becomes immediately effective. If a tool path is already drawn, the tool path is cleared when new parameter values are set.

- Graphic coordinate system

Select a desired graphic coordinate system for tool path drawing then set the corresponding number.

M

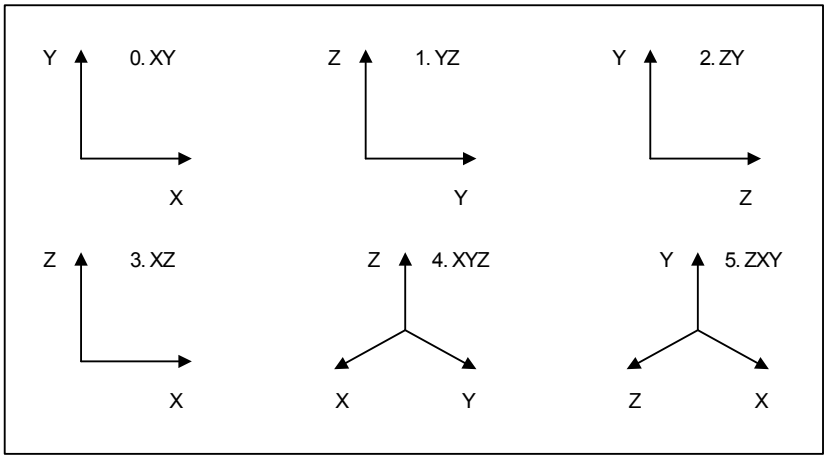


Fig. 13.1 (i) Graphic coordinate system (M series)

T

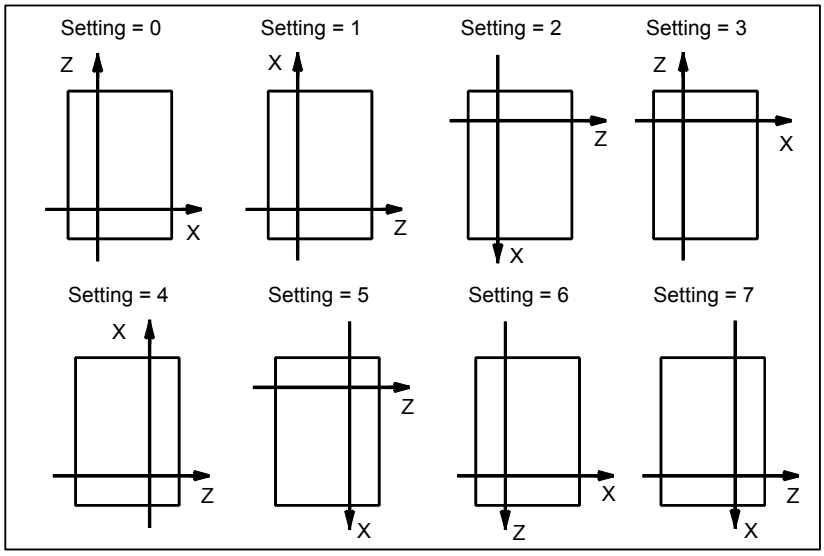


Fig. 13.1 (j) Graphic coordinate system (T series)

M

- Horizontal rotation angle

When a 3-dimensional graphic coordinate system such as 4.XYZ or 5.ZXY is selected, the coordinate system can be rotated with the horizontal plane used as the rotation plane. Set a rotation angle from -360° to +360°.

The initial rotation angle can be set with parameter No. 24831, and the actual rotation angle is equal to the "initial rotation angle + specified rotation angle".

In Fig. 13.1 (k) below, the graphic coordinate system XYZ is converted to X"Y"Z" by the following settings:

Initial rotation angle: 180°
Horizontal rotation angle: 30°

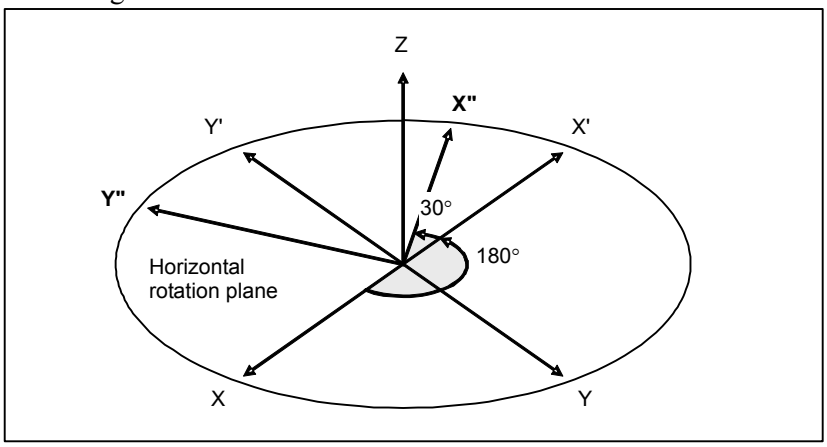


Fig. 13.1 (k) Coordinate system rotation in horizontal direction

- Vertical rotation angle

When a 3-dimensional coordinate system such as 4.XYZ or 5.ZXY is selected, the coordinate system can be rotated with an axis on the horizontal plane specified as a vertical rotation axis. Set a rotation angle from -360° to +360°.

A vertical rotation axis can be set with the angle with the horizontal axis of the screen on the horizontal plane. This angle can be set with parameter No. 24832.

In Fig. 13.1 (l) below, the graphic coordinate system XYZ is converted to X'Y'Z' by the following settings:

Rotation angle of the vertical rotation axis: 65°
Vertical rotation angle: 20°

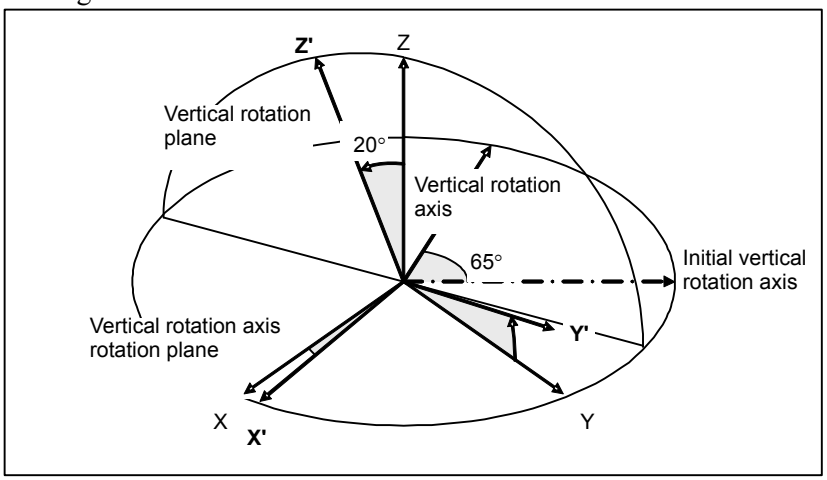


Fig. 13.1 (l) Coordinate system rotation in vertical direction

- Graphic color

Set a graphic color number for a tool path for each of cutting feed and rapid traverse.

1: Red, 2: Green, 3: Yellow, 4: Blue, 5: Purple, 6: Sky blue, 7: White

- Graphic range setting

Set a graphic range so that a tool path can be drawn in the tool path graphic area. Two methods are available:

1. Method that sets graphic center coordinates and a scale
2. Method that sets the maximum values and minimum values of a graphic range

Whether method 1 or method 2 is used is determined by which parameters were set most recently. A set graphic range is preserved even if the power is turned off.

1. Method that sets graphic center coordinates and a scale

Set the coordinates of the center of a tool path graphic area in the workpiece coordinate system. Then, set a scale used to contain the graphic range in the tool path graphic area.
Set a scale value from 0.01 to 100 (magnification).
As a smaller scale is set, a tool path can be drawn in a wider range. As a larger scale is set, a tool path is drawn by enlarging the tool path around the graphic center coordinates.

2. Method that sets the maximum values and minimum values of a graphic range

Set the maximum coordinates and minimum coordinates of a desired graphic range in the workpiece coordinate system. Drawing is performed so that the specified entire range is contained in the tool path graphic area.

From set maximum values and minimum values, graphic center coordinates and a sale are automatically calculated to update the graphic center coordinates and scale on the graphic parameter screen.
When a scale is automatically determined, the scale is clamped to within the range 0.01 to 100. Moreover, a maximum value must be greater than the corresponding minimum value.

NOTE
When the maximum values and minimum values of a graphic range are set, the graphic center coordinates and scale are automatically updated. However, when the graphic center coordinates and scale are changed, the maximum values and minimum values of the graphic range are not automatically updated.

- Automatic erasure

Before drawing is started, the previous drawing can be erased automatically.
1: Immediately before drawing is started, the previous drawing is erased automatically.
0: The previous drawing is not erased automatically.

- Graphic axis number

Set which controlled axis to be assigned to which graphic axis.
For each controlled axis, set one of the following graphic axis numbers:

First graphic axis:	1
Second graphic axis:	2
Third graphic axis:	3
Axis not used for drawing:	0

NOTE
1 When 0 is set for all controlled axes, it is assumed that 1, 2, and 3 are set sequentially for the first to the third controlled axes.
2 With the T series, a tool path is drawn along the first and second graphic axes. No tool path is drawn along the third graphic axis.

Operation for graphic parameter setting

Operation

- Moving the cursor

The cursor can be moved to a desired parameter by the page key 

or 

and the cursor key , , , or .
With the cursor keys, however, you cannot move from page 1 or 2 to page 3.

- Input of settings (absolute input)

Method 1
(1) Key in a value to be set.
(2) Press the [INPUT] soft key.
Method 2
(1) Key in a value to be set.
(2) Press the 

key.

- Input of settings (incremental input)

Method 1
(1) Key in a value to be incremented to or decremented from the current setting.
(2) Press the [+INPUT] soft key.

Procedure for tool path drawing

Procedure

- Start of drawing

(1) Display the tool path graphic screen.
(2) Press the [START] soft key.
The state that enables the movement of the tool in automatic operation or manual operation to be drawn is set.
Afterwards, tool path drawing continues even if another screen is displayed.
(3) Start automatic operation or manual operation.

- End of drawing

(1) Display the tool path graphic screen.
(2) Press the [END] soft key.
Tool path drawing ends.

- Erasure of the drawing

(1) Press the [END] soft key to end drawing.
(2) Press the [CLEAR] soft key. The tool path drawn so far is erased.

NOTE
1 Set the machine lock state to perform drawing only without moving the tool.
2 When the feedrate is high, the tool path may not be drawn correctly. In such a case, decrease the feedrate by performing, for example, a dry run.

Enlarged/reduced display

On the tool path graphic screen, you can move the center position of the tool path drawing or enlarge the tool path drawing while viewing the drawn tool path.
If any of these operations is executed, the tool path already drawn is cleared.

- Procedure for changing the graphic range by setting a graphic center and magnification

The center position of drawing can be moved. At the same time, the scale can also be changed. So, the tool path can be enlarged or reduced at a desired new center position.

(1) Press the [SCALE] soft key then the [CENTER] soft key.
A yellow cursor appears at the center of the screen, and the soft key display is changed.
(2) Move the yellow cursor to a new graphic center position by using the cursor key , , , or .
(3) When changing the scale, key in a value from 0.01 to 100 (magnification) then press the [INPUT] soft key. An input value is displayed at "SCALE" in the lower-right corner of the screen.
When you press the [+INPUT] soft key, the current magnification is incremented by an input value.
(4) Press the [EXEC] soft key to end the operation.
After this step, the setting for graphic movement is effective to enable drawing with the new setting.

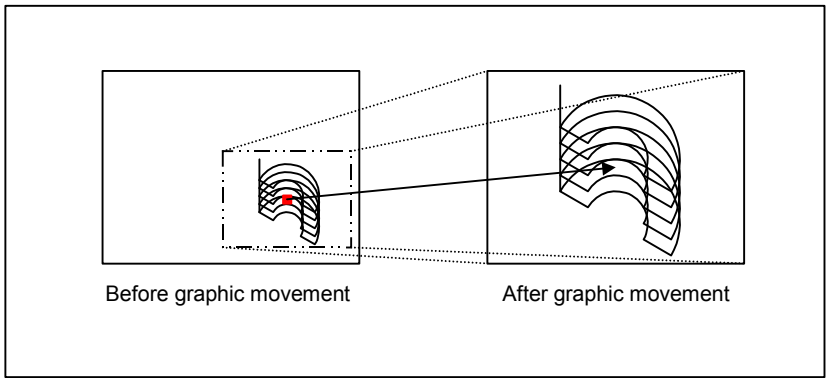


Fig. 13.1 (m) Graphic movement (magnification = 2.00)

- Procedure for changing the graphic range with a rectangle

A tool path can be drawn by enlarging a specified rectangular area.

(1) Press the [SCALE] soft key then the [RECTANGLE] soft key.
Two cursors, one in red and the other in yellow, appear at the center of the screen, and the soft key display is changed.

(2) Move the yellow cursor by using the cursor key , , , or . The cursor to be moved can be switched by pressing the [CURSOR UP/DOWN] soft key.
Move the two cursors to the diagonal points of a new rectangular graphic range. A tool path is drawn next time so that the drawn tool path is contained in this rectangular range.

(3) Press the [EXEC] soft key to end the operation.
After this step, the setting of graphic enlargement is effective to enable drawing with the new setting.

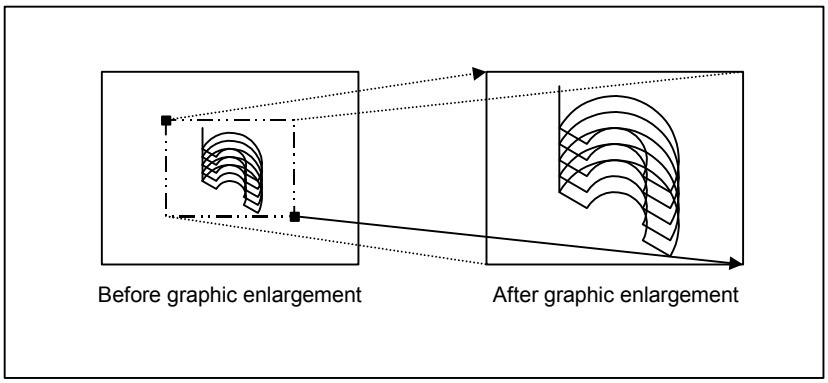


Fig. 13.1 (n) Graphic enlargement

NOTE
1 To stop an enlargement/reduction operation, press the [CANCEL] soft key.
2 Even if you perform an enlargement/reduction operation, the tool path already drawn on the screen is neither moved nor enlarged. The setting for enlargement/reduction becomes effective starting with tool path drawing after you press the [EXEC] soft key.

13.2 DYNAMIC GRAPHIC DISPLAY

Overview

The dynamic graphic display function has two features:

- Path Drawing
  The path of coordinates specified in a program is drawn on the screen.
  By displaying a travel path on the screen, the path can be checked easily before performing machining actually.
- Animation
  The figure of a workpiece to be machined by a programmed tool movement is drawn.
  By drawing the 3-dimensional figure of a workpiece to be machined in an animation-like way, the intermediate machining process and final figure can be grasped easily.

When machining is performed according to a program, this function can draw a tool path with another program.
This function performs drawing much faster than the graphic display function based on automatic operation, so that a program check can be made more swiftly.
This function is distinguished from automatic program operation as indicated by the following terminology:

Automatic operation	Operation performed for actual machining
Background operation	Virtual operation performed for drawing

13.2.1 Path Drawing

Overview

The following tool path drawing screens are used to make various settings and execute drawing:

- GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen
  This screen is used to set data needed for tool path drawing.
- PATH GRAPHIC screen
  This screen is used for tool path drawing.
- PATH GRAPHIC (TOOL POSITION) screen
  This screen enables the position of the tool during operation to be checked on the drawn tool path.

13.2.1.1 GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen

This screen is used to set graphic parameters needed for tool path drawing.
Data set using this screen is made valid by displaying the PATH GRAPHIC screen or executing drawing.
If a tool path is already drawn, the path is erased.
Graphic parameter data once set is preserved even if the power is turned off.

Graphic Parameter (Dynamic Graphic) Screen Procedure

Procedure

1 Press the function key 

(or 

when a small MDI unit is used) to display the GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen.

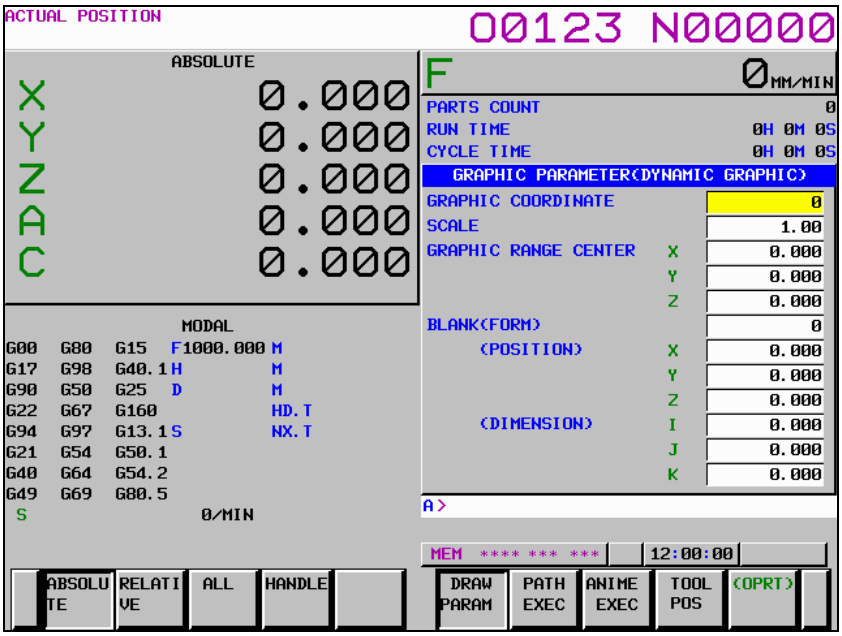


Fig. 13.2.1.1 (a) GRAPHIC PARAMETER screen (first page)

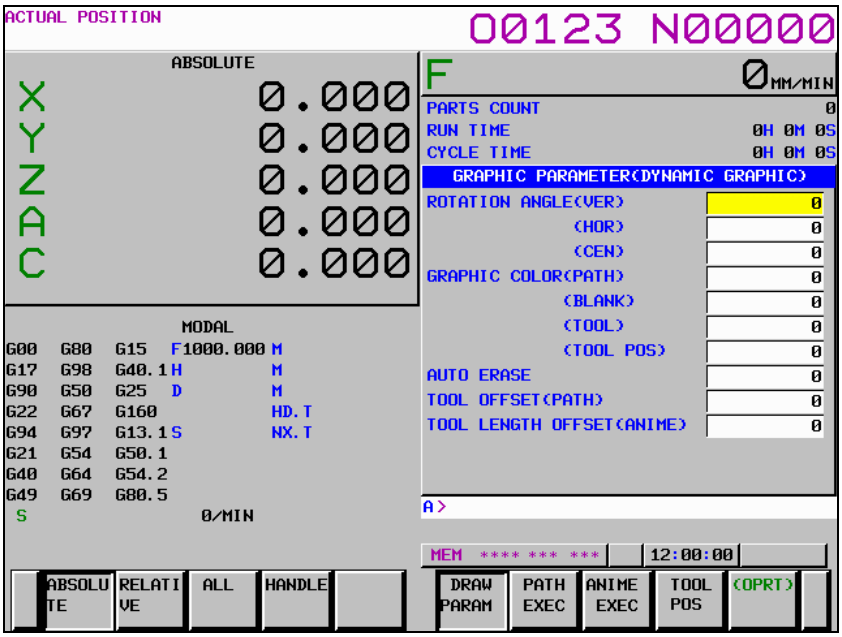


Fig. 13.2.1.1 (b) GRAPHIC PARAMETER screen (second page)

2 Two screens are used for the GRAPHIC PARAMETER screen.
Use the MDI page keys to switch between the screens for display of a desired setting item.

3 Move the cursor to a desired setting item with the MDI cursor keys then key in a desired value with the numeric keys.

4 For direct input of the value set in step 3, press the 

key or the [INPUT] soft key.
For incremental input of the value set in step 3, press the [+INPUT] soft key.

For each setting item, see the explanation.

Explanation

The setting items on the GRAPHIC PARAMETER screen are described below.

- Graphic coordinate system

Select a graphic coordinate system for drawing from the following and set its number.

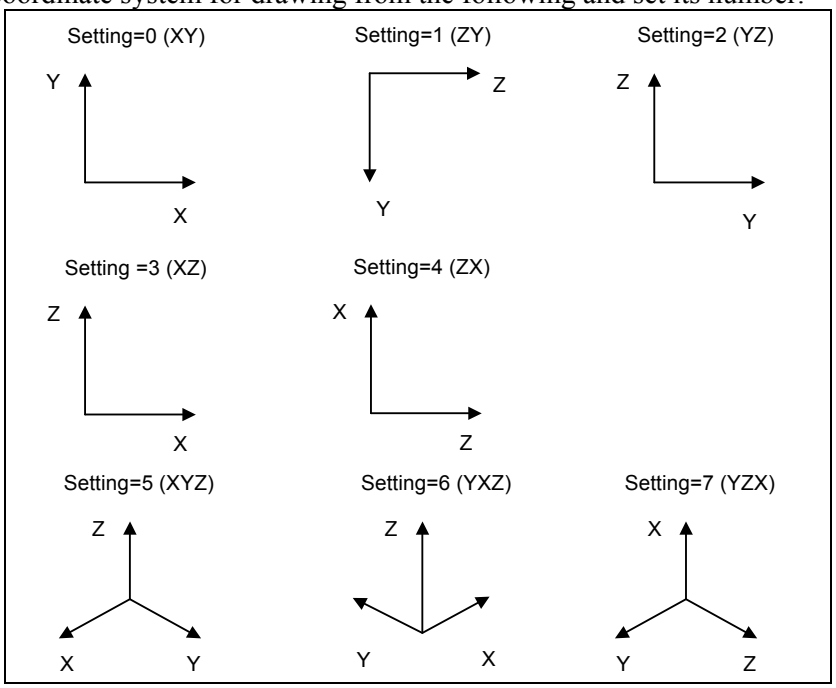


Fig. 13.2.1.1 (c) Graphic coordinate system

- Scale

Set the scale factor for drawing in the range of 0.01 to 100.00 (times).
With a small scale factor, it is possible to draw within a wide range.
With a large scale factor, it is possible to draw in the vicinity of the graphic center being enlarged.

- Graphic range center

To specify the center coordinates of a graphic range, set a coordinate on each axis in the workpiece coordinate system of the program executed for drawing.

NOTE
Set a coordinate on each axis in the machine coordinate system in case of bit 3 (BGM) of parameter No.11329=1.

- Blank figure

With a drawing program, set the figure, position, and dimensions of a blank to be machined.
The graphic range where the blank figure is included in the drawing area is automatically decided by the value set here.

NOTE
1 With a lathe system, no rectangular parallelepiped can be set.
2 With a lathe system, the Y-axis, one of the basic three axes, may not exist. In such a case, the Y-axis for the reference position of a blank need not be set.
3 When a figure or dimension is set, a scale value and screen center coordinates are automatically set as the initial graphic range values so that the blank figure is located within the graphic range on the screen. Until a modification is made to the blank figure, dimension or the setting of the graphic coordinate system, the graphic range is determined by the set scale and screen center coordinates.

Figure

Select a type of blank figure from the following (Table 13.2.1.1 (a)) and set the corresponding value:

Table 13.2.1.1 (a)

Setting	Figure
0	Column or cylinder (parallel with the Z-axis)
1	Rectangular parallelepiped

Position

Set the reference position of a blank with coordinates (X,Y,Z) in the workpiece coordinate system.

NOTE
Set the reference position of a blank with coordinates (X,Y,Z) in the machine coordinate system in case of bit 3 (BGM) of parameter No.11329=1.

Dimensions

Set the dimensions of each type of blank figure as indicated below (Table 13.2.1.1 (b)).

Table 13.2.1.1 (b)

Type of blank figure	Dimension I	Dimension J	Dimension K
Rectangular parallelepiped	Length in X-axis direction	Length in Y-axis direction	Length in Z-axis direction
Column	Diameter of column	0	Length of column
Cylinder	Diameter of outer circle of cylinder	Diameter of inner circle of cylinder	Length of cylinder

The relationship between the position and dimensions of a blank is shown below.

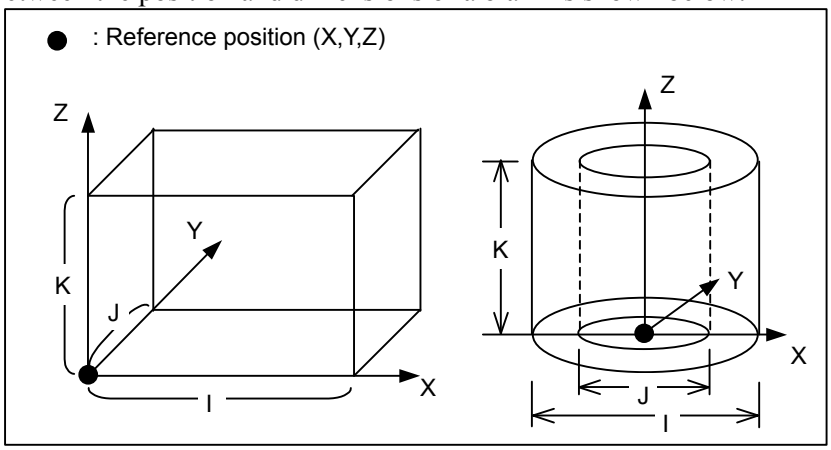


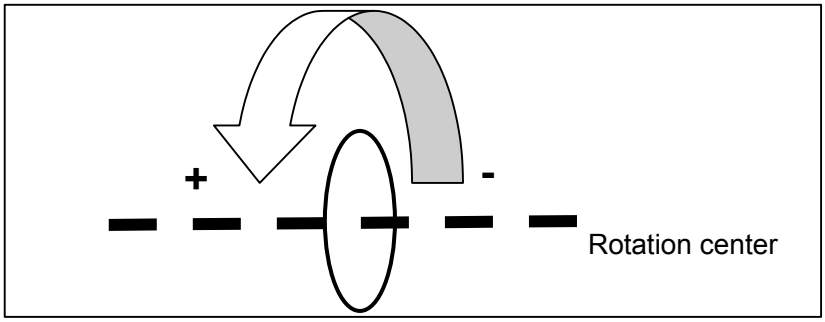
Fig. 13.2.1.1 (d)

- Rotation angle

Set a rotating angle of the graphic coordinate system that centers on the graphic range center. The rotating angle is a range of -360°-+360°.
Set a rotating angle as a reference position (position of the rotating angle 0°) on the indicated direction of each graphic coordinate system.

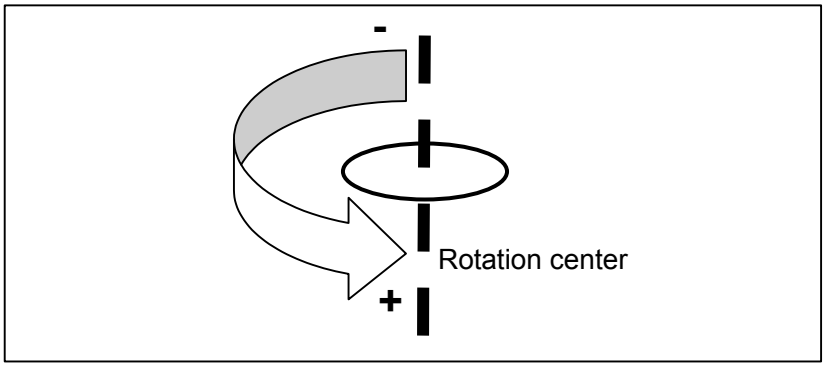
Vertical plane rotation angle

Set a rotating angle at the horizontal direction center in front of the screen.
The rotation direction is as follows.



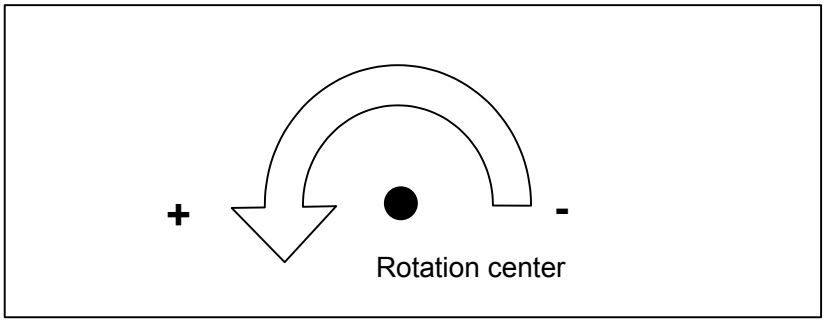
Horizontal plane rotation angle

Set a rotating angle at the vertical direction center in front of the screen.
The rotation direction is as follows.



Screen center rotation angle

Set a rotating angle at the vertical direction center of the screen plane.
The rotation direction is as follows.



- Graphic color

Set colors to be used for tool path drawing.
The colors that can be set are indicated below together with their setting values:

Graphic color	White	Red	Green	Yellow	Blue	Purple	Light blue	White
Setting value	0	1	2	3	4	5	6	7

Path

Set colors to be used for a drawn tool path.

Tool position

Set a graphic cursor color to be used for PATH GRAPHIC (TOOL POSITION) screen.

- Auto erase

When drawing is started with the [START] soft key from a state where drawing is not being executed or is not stopped temporarily, the previously drawn path can be erased.

Setting 0: The previously drawn path is not erased.
1: The previously drawn path is erased.

- Tool offset(Path)

For tool path drawing, whether to enable or disable the tool offset function (tool length compensation, cutter/tool-noise radius compensation) can be selected.

Setting 0: The tool offset function is enabled for drawing.
1: The tool offset function is disabled for drawing.

13.2.1.2 PATH GRAPHIC screen

The PATH GRAPHIC screen is used to draw a tool path. The following operations can be performed:

- Starting/ending tool path drawing
- Rewind of a drawing target program
- Erasing a drawn tool path

The screen is composed of the following items:

(1) Drawing area
(2) Status display of Background operation
(3) Program name and sequence number for drawing execution
(4) Current coordinates (absolute coordinates)
(5) Feedrate and M/S/T/D code instruction information
(6) Graphic coordinate system

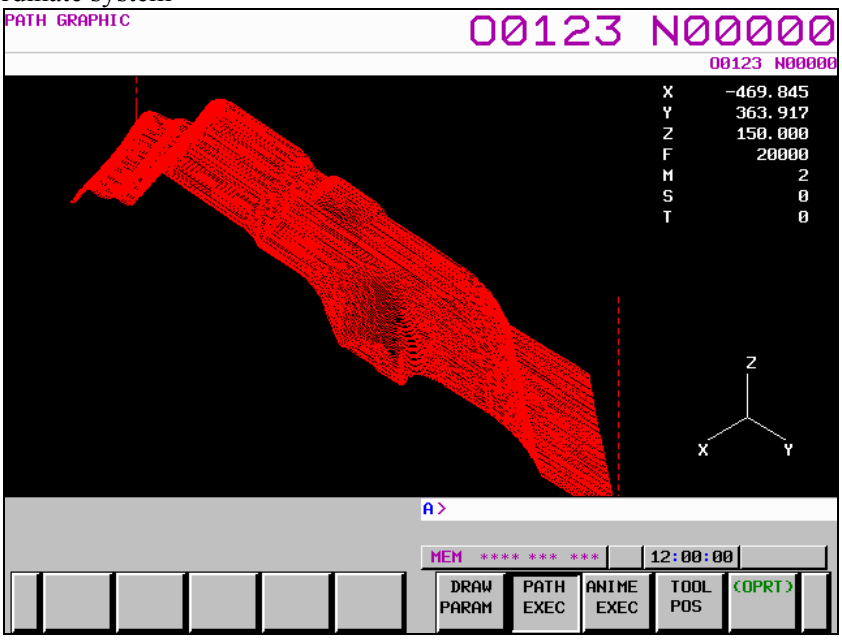


Fig. 13.2.1.2 (a) PATH GRAPHIC screen

Path Graphic Screen Procedure

Procedure

1 Press the function key  (or  when a small MDI unit is used) to display the GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen.

2 Press the [PATH EXEC] soft key. The PATH GRAPHIC screen is displayed.

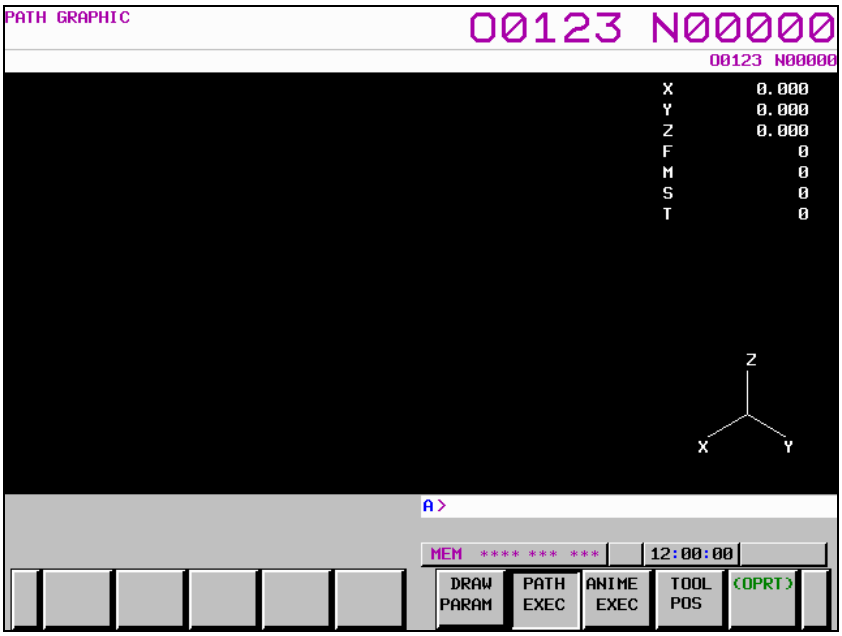


Fig. 13.2.1.2 (b) PATH GRAPHIC screen

3 Press the [(OPRT)] soft key. The soft keys for tool path drawing are displayed.



Fig. 13.2.1.2 (c) PATH GRAPHIC screen (operation)

4 Press the continuous menu key ▷ to display the soft keys for enlarging/reducing/moving the graphic range.



Fig. 13.2.1.2 (d) PATH GRAPHIC screen (enlarging/reducing/moving the graphic range)

5 Press the [COORDINATES] soft key to display the soft keys for changing the graphic coordinate system.

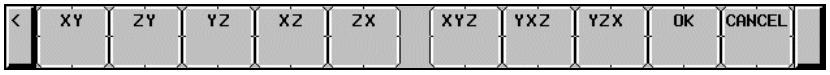


Fig. 13.2.1.2 (e) PATH GRAPHIC screen (changing the graphic coordinate system)

6 Press the [ROTATION] soft key to display the soft keys for rotating the graphic coordinate system.



Fig. 13.2.1.2 (f) PATH GRAPHIC screen (rotating the graphic coordinate system)

For the operation of each soft key, see the explanation.

Explanation

- Graphic program selection

Press the [PRGRAM SELECT] soft key to select a drawing target program.
Pressing the [PRGRAM SELECT] soft key displays the program list screen.
On the program list screen, move the cursor to a desired program then press the [DRAW SELECT] soft key.



Fig. 13.2.1.2 (g) Program list screen ([DRAW SELECT] soft key)

Pressing the [DRAW SELECT] soft key selects a drawing target program and switches the screen display to the PATH GRAPHIC screen.

The file whose name is prefixed by a "#" mark on the program list screen is selected as the drawing target program. And the mark of "+" is displayed in the file that has been selected as a program for drawing in other path systems.

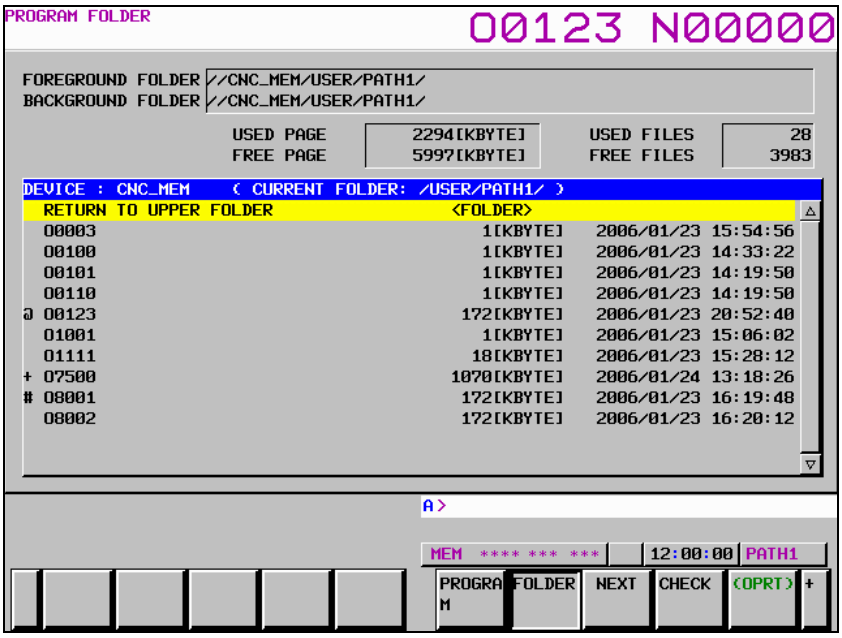


Fig. 13.2.1.2 (h) Program list screen (drawing target program selection state)

NOTE
The file that can be selected as the drawing target program is only a file that can be selected as the main program.

- Rewind of a drawing target program

If the execution of drawing of a selected program has ended or is stopped halfway, press the [REWIND] soft key to restart drawing from the beginning of the program.

- Starting / Stopping drawing

To draw the tool path of a program selected for drawing, press one of the following soft keys displayed by step 3 mentioned above:

- [START] soft key
  This soft key starts drawing from the head of the program.
- [1BLOCK] soft key
  This soft key executes the program for drawing and stops temporarily on a block-by-block basis as with ordinary single block stop operation.

If a program is executed with a soft key above, the soft key display is changed to the following:

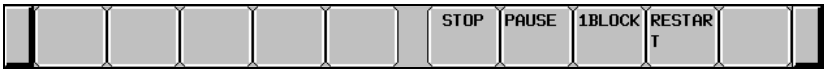


Fig. 13.2.1.2 (i) Soft keys displayed during drawing execution

The operations of these soft keys are as follows:

- [STOP] soft key
  This soft key terminates the execution of the drawing target program to stop drawing.
- [PAUSE] soft key
  This soft key temporarily stops the execution of the drawing target program to stop drawing temporarily.
- [1BLOCK] soft key
  This soft key executes the program for drawing and stops drawing temporarily on a block-by-block basis as with ordinary single block stop operation.
- [RESTART] soft key
  If the [RESTART] soft key is pressed in the stop state set by the [PAUSE] soft key or the [1BLOCK] soft key, the execution of drawing can be restarted from the block where drawing is stopped.

NOTE
When the new path was drawn by the operation of starting drawing without erasing of the old path before the operation, it is impossible to redraw the old path by each operation of enlarging/reducing/moving the graphic range and changing/rotating the graphic coordinate system.

The state of drawing is indicated as follows:

DRAWING: Indicates that drawing is being executed.

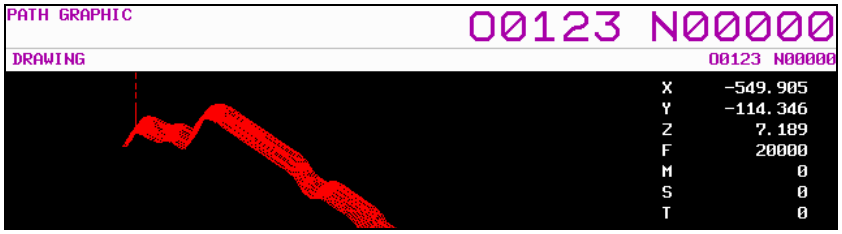


Fig. 13.2.1.2 (j) State indication during drawing

STOP: Indicates that drawing is temporarily stopped.

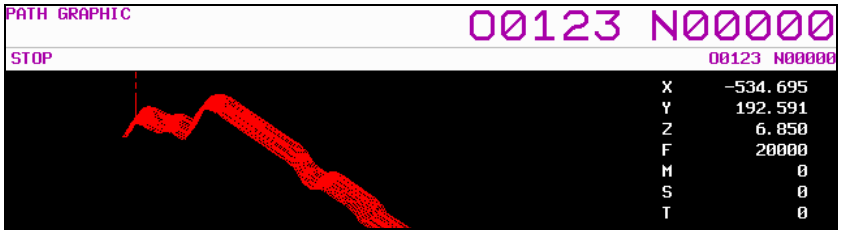


Fig. 13.2.1.2 (k) State indication during temporary stop

ALM: Indicates that occurring of alarm in the Background operation.

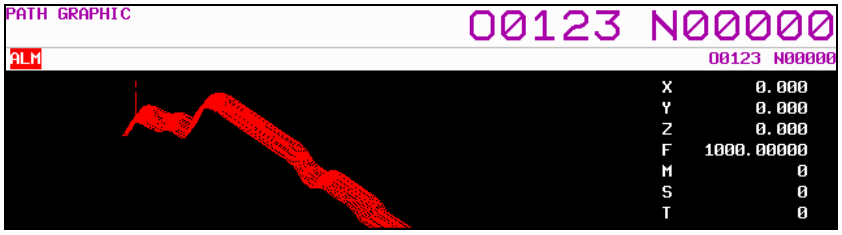


Fig. 13.2.1.2 (l) State indication during occurring of alarm

- End of drawing

When M02 or M30 is executed, the program executed for drawing terminates drawing.
Upon program termination, the soft key display returns to the soft keys (Fig. 13.2.1.2 (c)) displayed before drawing is started.

- Erasing a drawn tool path

Press the [ERASE] soft key to erase a drawn tool path.

NOTE
1 If the screen display is switched or the path is switched during tool path drawing, the background operation is stopped to end drawing.
2 A tool path once drawn is erased when the screen display is switched or the path is switched.

- Enlarging/reducing the graphic range

The following soft keys displayed by step 4 are used:

- [LARGE] soft key
  This soft key increases the scale to enlarge the graphic range.
- [SMALL] soft key
  This soft key decreases the scale to reduce the graphic range.
- [AUTO] soft key
  When the blank figure specified in a graphic parameter is input, this soft key automatically scales the graphic range so that the blank figure is contained within the graphic range.
  If no blank figure is input, this soft key has no effect.

NOTE
1 Set the unit of scale for one enlargement/reduction operation in parameter No. 14713.
2 An enlargement/reduction scale used here is set in the graphic parameter for scale.

- Moving the graphic range

The following soft keys displayed by step 4 are used:

- [←MOVE] soft key
  This soft key moves the graphic range to left.
- [MOVE→] soft key
  This soft key moves the graphic range to right.
- [↑MOVE] soft key
  This soft key moves the graphic range upward.
- [↓MOVE] soft key
  This soft key moves the graphic range downward.
- [CENTER] soft key
  This soft key returns the graphic range to the original position.

NOTE
1 Set the travel increment made by one horizontal move operation in parameter No. 14714.
2 Set the travel increment made by one vertical move operation in parameter No. 14715.
3 The graphic range modified here is not set in the graphic parameter for graphic range center.

- Changing the graphic coordinate system

The following soft keys displayed by step 5 are used.
A graphic coordinate system selected here is the same one as set in the graphic parameter for the graphic coordinate system.

- [XY] soft key
  This soft key selects the graphic coordinate system of XY (with a setting of 0).
- [ZY] soft key
  This soft key selects the graphic coordinate system of ZY (with a setting of 1).
- [YZ] soft key
  This soft key selects the graphic coordinate system of YZ (with a setting of 2).
- [XZ] soft key
  This soft key selects the graphic coordinate system of XZ (with a setting of 3).
- [ZX] soft key
  This soft key selects the graphic coordinate system of ZX (with a setting of 4).
- [XYZ] soft key
  This soft key selects the graphic coordinate system of XYZ (with a setting of 5).
- [YXZ] soft key
  This soft key selects the graphic coordinate system of YXZ (with a setting of 6).
- [YZX] soft key
  This soft key selects the graphic coordinate system of YZX (with a setting of 7).
- [OK] soft key
  This soft key changes the current graphic coordinate system to the graphic coordinate system selected by one of the soft keys above.
- [CANCEL] soft key
  This soft key cancels the graphic coordinate system selected by one of the soft keys above to return to the original graphic coordinate system.

NOTE
The graphic coordinate system selected here is set in the graphic parameter for the graphic coordinate system.

- Rotating the graphic coordinate system

The following soft keys displayed by step 6 are used.

- [↑] soft key
  This soft key rotates the graphic coordinate system upward.
- [↓] soft key
  This soft key rotates the graphic coordinate system downward.
- [←] soft key
  This soft key rotates the graphic coordinate system to left.
- [→] soft key
  This soft key rotates the graphic coordinate system to right.
- [CW] soft key
  This soft key rotates the graphic coordinate system clockwise.
- [CCW] soft key
  This soft key rotates the graphic coordinate system counterclockwise.
- [OK] soft key
  This soft key changes the rotation angle of the current graphic coordinate system to the one set by one of the soft keys above.
- [CANCEL] soft key
  This soft key cancels the rotation of the graphic coordinate system performed by one of the soft keys above to return to the original graphic coordinate system.

NOTE
1 Set the travel increment made by one rotation operation in parameter No. 14716.
2 The rotation angle of the graphic coordinate system set here is not set in the graphic parameter for rotation angle.

13.2.1.3 PATH GRAPHIC (TOOL POSITION) screen

The tool position during operation can be checked by displaying the cursor for indicating the tool position during operation on the tool path drawn on the PATH GRAPHIC (EXECUTION) screen.

The screen is composed of the following items:
(1) Drawing area
(2) Current coordinates (absolute coordinates)
(3) Feedrate and M/S/T/D code instruction information
(4) Graphic coordinate system
(5) Cursor to show tool position

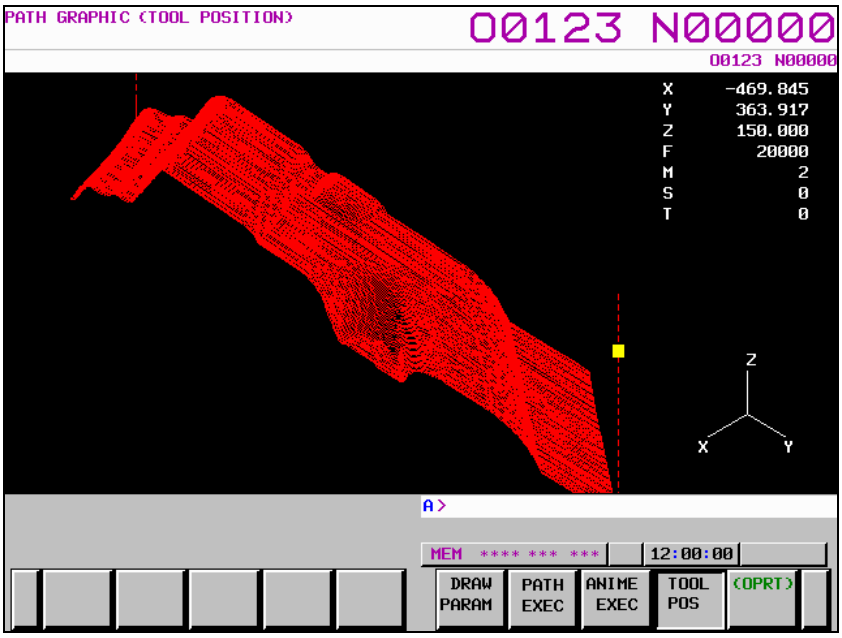


Fig. 13.2.1.3 (a) PATH GRAPHIC (TOOL POSITION) screen

Path Graphic (Tool Position) Screen Procedure

Procedure

1 Press the function key  (or  when a small MDI unit is used) to display the PATH GRAPHIC (PARAMETER) screen.

2 Press the [TOOL POS] soft key.
The screen display changes to the PATH GRAPHIC (TOOL POSITION) screen and the cursor indicating the tool position is displayed.

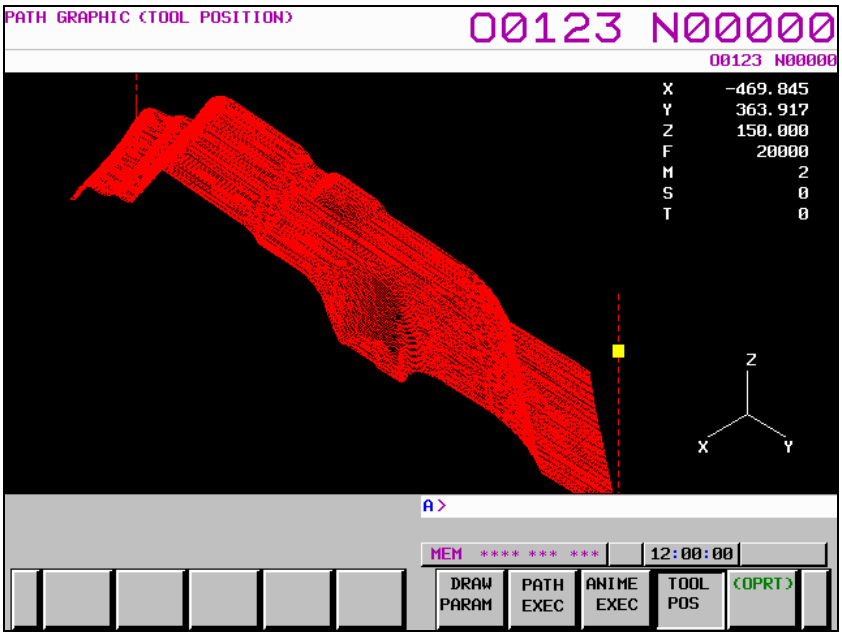


Fig. 13.2.1.3 (b) PATH GRAPHIC (TOOL POSITION) screen

For the method of checking the current tool position, see the explanation.
Pressing a soft key other than the [TOOL POS] soft key displays the corresponding screen.

Explanation

Use the following procedure to check the tool position during operation on the PATH GRAPHIC (TOOL POSITION) screen:

(1) Select a drawing target program for operation.
(2) Draw the tool path of the selected program on the PATH GRAPHIC screen.
(3) After the end of drawing, switch the screen display to the PATH GRAPHIC (TOOL POSITION) screen.
(4) Start automatic operation with the program executed for tool path drawing.

After automatic operation is started, the blinking cursor moves along the drawn tool path as the tool moves. The cursor blinking rate is fast when the tool is moving, and the cursor blinking rate is slow when the tool is stopped.

The following items displayed on the screen are provided for the program under automatic operation:

- Current coordinates
- Feedrate and M/S/T/D code specification information

NOTE

1 A tool path drawn by setting the tool offset parameter to 1 (to disable the tool offset function) is different from the actual tool path. In this case, the cursor indicating the tool position may not move along the drawn tool path.
2 The tool path cannot be drawn correctly if a command not supporting drawing or a command that specifies operation different from operation performed during drawing is specified or if an operation dependent on the operation state or setting on the machine side is performed. So, the tool path in actual operation may differ from a drawn tool path. In this case, the cursor indicating the tool position does not move along the drawn tool path.

NOTE
3 When the graphic parameter of the graphic coordinate system, scale, graphic range center, blank figure / position / dimensions and rotation angle is changed, the drawn tool path drawn is erased.
Therefore, please draw the tool path again on the PATH GRAPHIC screen to display the tool position by setting the changed graphic parameter.

13.2.2 Animation

Overview

For animation drawing, make necessary settings and perform operations for drawing execution on the following screens:

- GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen
  On this screen, data required to execute animation drawing can be set.
- ANIMATION GRAPHIC screen
  This screen is used for animation drawing.
  On this screen, the graphic range can be reduced/enlarged, and the graphic coordinate system can be rotated.

13.2.2.1 GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen

This screen is used to set graphic parameters needed for animation drawing.
Data set using this screen is made valid by displaying the ANIMATION GRAPHIC screen or executing drawing.
Graphic parameter data once set is preserved even if the power is turned off.

Graphic Parameter (Dynamic Graphic) Screen Procedure

Procedure

1 Press the function key GRAPH (or CSTM GRAPH when a small MDI unit is used) to display the GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen.

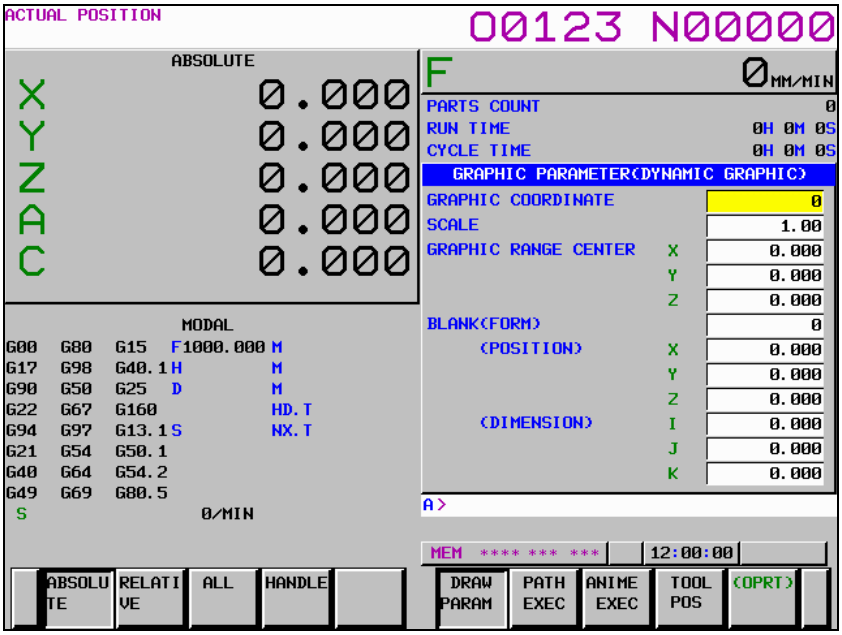


Fig. 13.2.2.1 (a) GRAPHIC PARAMETER screen (first page)

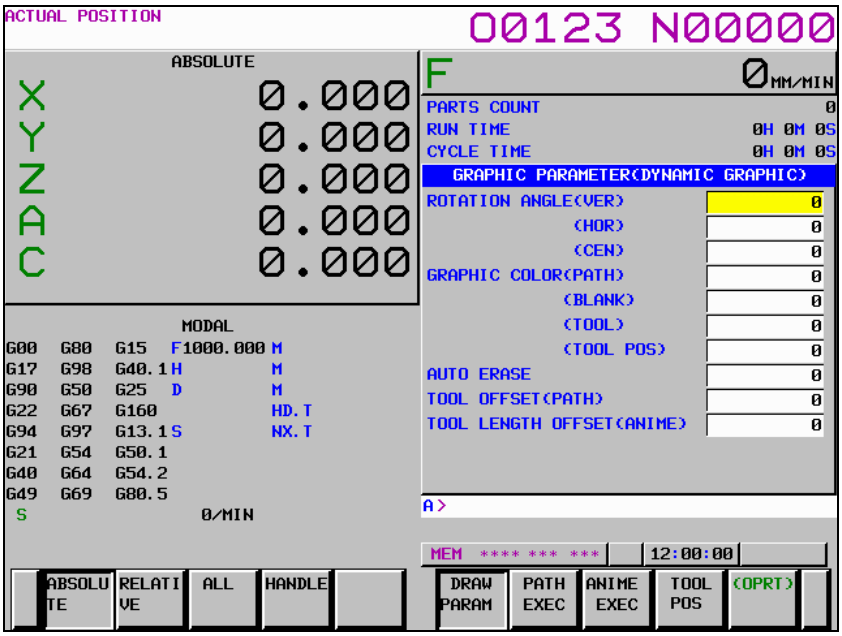


Fig. 13.2.2.1 (b) GRAPHIC PARAMETER screen (second page)

2 Two screens are used for the GRAPHIC PARAMETER screen.
Use the MDI page keys to switch between the screens for display of a desired setting item.

3 Move the cursor to a desired setting item with the MDI cursor keys then key in a desired value with the numeric keys.

4 For direct input of the value set in step 3, press the INPUT key or the [INPUT] soft key.
For incremental input of the value set in step 3, press the [+INPUT] soft key.

For each setting item, see the explanation.

Explanation

The setting items on the GRAPHIC PARAMETER screen are described below.
However, the graphic parameters listed below are shared for tool path drawing. So, refer to the explanation of the GRAPHIC PARAMETER screen for tool path drawing.

- Graphic coordinate system
- Scale
- Graphic range center
- Rotation angles (vertical plane, horizontal plane, screen center)

- Blank figure

With a drawing program, set the figure, position, and dimensions of a blank to be machined.

NOTE
1 With a lathe system, no rectangular parallelepiped can be set.
2 With a lathe system, the Y-axis, one of the basic three axes, may not exist. In such a case, the Y-axis for the reference position of a blank need not be set.
3 When a figure or dimension is set, a scale value and screen center coordinates are automatically set as the initial graphic range values so that the blank figure is located within the graphic range on the screen. Until a modification is made to the blank figure, dimension or the setting of the graphic coordinate system, the graphic range is determined by the set scale and screen center coordinates.

Figure

Select a type of blank figure from the following and set the corresponding value:

Setting	Figure
0	Column or cylinder (parallel with the Z-axis)
1	Rectangular parallelepiped

Position

Set the reference position of a blank with coordinates (X,Y,Z) in the workpiece coordinate system.

NOTE
Set the reference position of a blank with coordinates (X,Y,Z) in the machine coordinate system in case of bit 3 (BGM) of parameter No.11329=1.

Dimensions

Set the dimensions of each type of blank figure as indicated below.

Type of blank figure	Dimension I	Dimension J	Dimension K
Rectangular parallelepiped	Length in X-axis direction	Length in Y-axis direction	Length in Z-axis direction
Column	Diameter of column	0	Length of column
Cylinder	Diameter of outer circle of cylinder	Diameter of inner circle of cylinder	Length of cylinder

The relationship between the position and dimensions of a blank is shown below (Fig. 13.2.2.1 (c)).

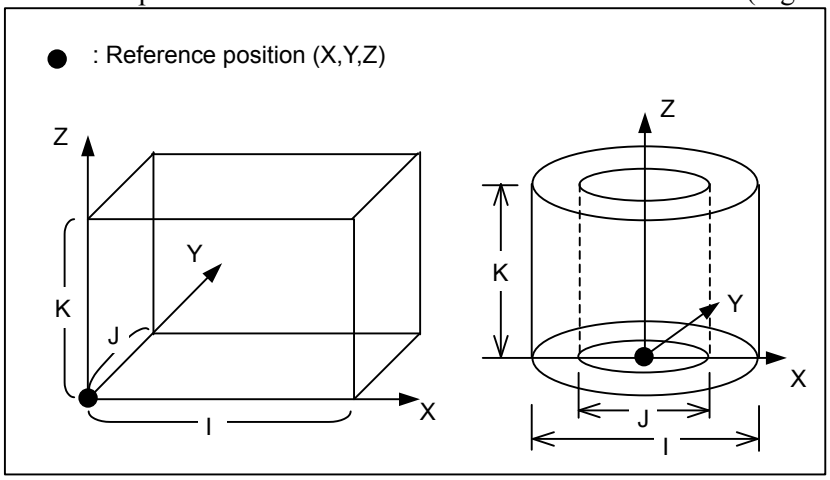


Fig. 13.2.2.1 (c)

- Graphic color

Set colors to be used for animation drawing.
The colors that can be set are indicated below together with their setting values:

Graphic color	White	Red	Green	Yellow	Blue	Purple	Light blue	White
Setting value	0	1	2	3	4	5	6	7

Blank

Set the color of a blank to be drawn in animation drawing.

Tool

Set the color of a tool to be drawn in animation drawing.

- Tool length offset (Anime)

For animation drawing, whether to enable or disable the tool length offset can be selected.
Setting 0: The tool length offset is disabled for drawing.
1: The tool length offset is enabled for drawing.

NOTE
In animation drawing, the cutter/tool-noise radius compensation is always enabled for drawing.

13.2.2.2 ANIMATION GRAPHIC screen

The ANIMATION GRAPHIC screen is used to draw a animation. The following operations can be performed:

- Starting/ending animation drawing
- Rewind of a drawing target program
- Initializing a blank
- Enlarging/reducing/moving the graphic range
- Rotating the graphic coordinate system

The screen configuration is as follows:

(1) Drawing area
(2) Status display of Background operation
(3) Program name and sequence number for drawing execution
(4) Current coordinates (absolute coordinates)
(5) Feedrate and M/S/T/D code instruction information
(6) Graphic coordinate system
(7) Material position, Material dimension, Scale, Graphic range center of graphic Parameter
(8) Tool name

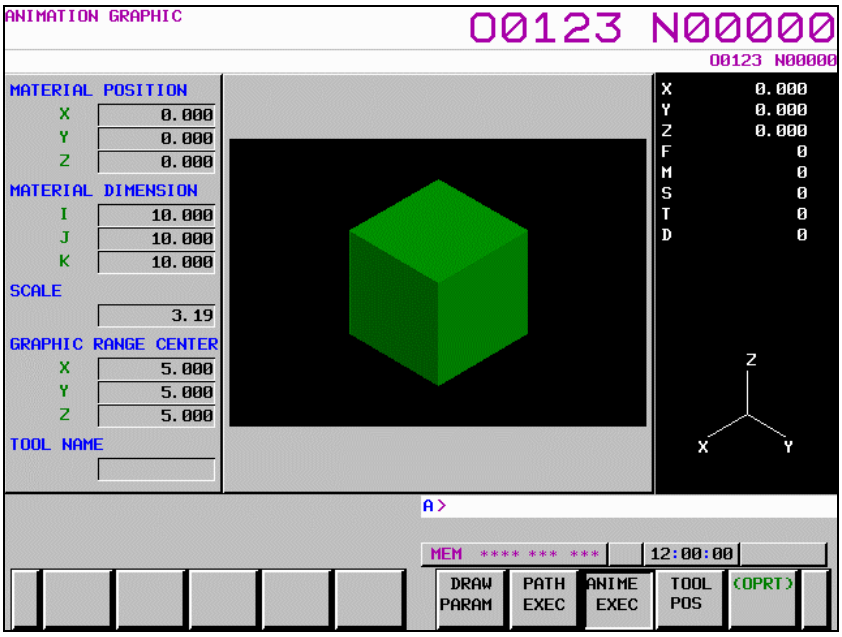


Fig. 13.2.2.2 (a) ANIMATION GRAPHIC screen

Animation Graphic Screen Procedure

Procedure

1 Press the function key 

(or 

when a small MDI unit is used) to display the GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen.

2 Press the [ANIME EXEC] soft key. The ANIMATION GRAPHIC screen is displayed.

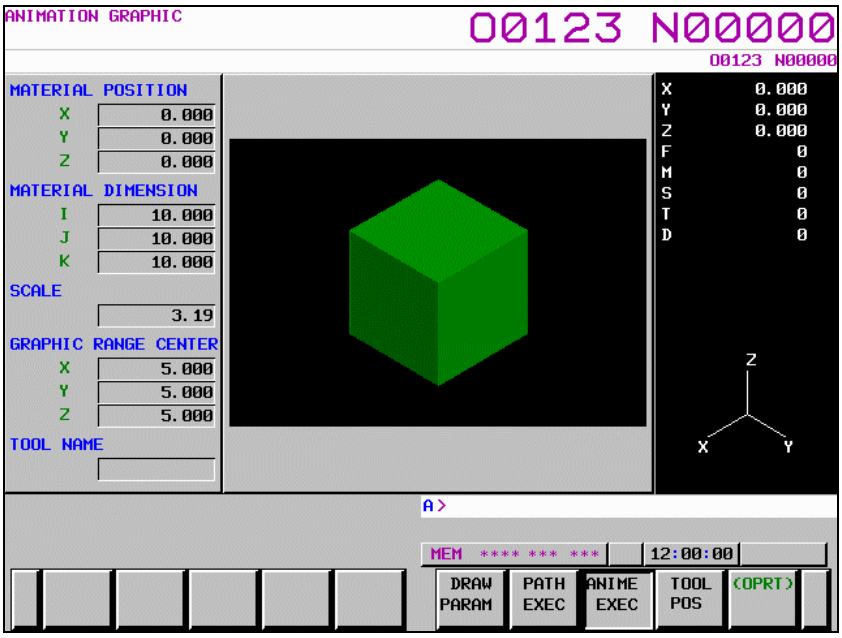


Fig. 13.2.2.2 (b) ANIMATION GRAPHIC screen

3 Press the [(OPRT)] soft key. The soft keys for tool path drawing are displayed.

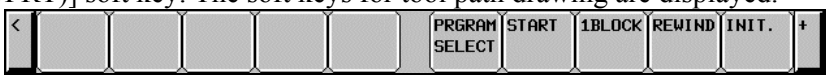


Fig. 13.2.2.2 (c) ANIMATION GRAPHIC screen (operation)

4 Press the continuous menu key  to display the soft keys for enlarging/reducing/moving the graphic range.



Fig. 13.2.2.2 (d) ANIMATION GRAPHIC screen (enlarging/reducing/moving the graphic range)

5 Press the [COORDINATES] soft key to display the soft keys for changing the graphic coordinate system.

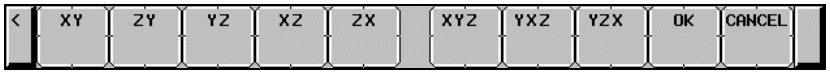


Fig. 13.2.2.2 (e) ANIMATION GRAPHIC screen (changing the graphic coordinate system)

6 Press the [ROTATION] soft key to display the soft keys for rotating the graphic coordinate system.



Fig. 13.2.2.2 (f) ANIMATION GRAPHIC screen (rotating the graphic coordinate system)

For the operation of each soft key, see the explanation.

Explanation

The operations listed below are the same operations as for the tool path drawing screen. See the explanation of the tool path drawing screen.

- Graphic program selection
- Rewind of a drawing target program
- Starting / Stopping of drawing
- End of drawing

- Initializing a blank

Press the [INIT.] soft key to initialize and return a drawn machining profile to the original blank figure.

NOTE
1 Blank initialization can also be performed by any of the following operations:
  - Start of drawing
  - Changing of the graphic coordinate system and graphic range by performing enlargement/reduction/movement/rotation operations
  - Change of screen
2 A machining profile once drawn for a blank is erased when the screen or the path is switched.

- Enlarging/reducing the graphic range

The following soft keys displayed by step 4 are used:

- [LARGE] soft key
  This soft key increases the scale to enlarge the graphic range.
- [SMALL] soft key
  This soft key decreases the scale to reduce the graphic range.
- [AUTO] soft key
  When the blank figure specified in a graphic parameter is input, this soft key automatically scales the graphic range so that the blank figure is contained within the graphic range.
  If no blank figure is input, this soft key has no effect.

NOTE
1 Set the unit of scale for one enlargement/reduction operation in parameter No. 14713.
2 An enlargement/reduction scale used here is set in the graphic parameter for scale.

- Moving the graphic range

The following soft keys displayed by step 4 are used:

- [←MOVE] soft key
  This soft key moves the graphic range to left.
- [MOVE→] soft key
  This soft key moves the graphic range to right.
- [↑MOVE] soft key
  This soft key moves the graphic range upward.
- [↓MOVE] soft key
  This soft key moves the graphic range downward.
- [CENTER] soft key
  This soft key returns the graphic range to the original position.

NOTE
1 Set the travel increment made by one horizontal move operation in parameter No. 14714.
2 Set the travel increment made by one vertical move operation in parameter No. 14715.
3 The graphic range modified here is not set in the graphic parameter for graphic range center.

- Changing the graphic coordinate system

The following soft keys displayed by step 5 are used.
A graphic coordinate system selected here is the same one as set in the graphic parameter for the graphic coordinate system.

- [XY] soft key
  This soft key selects the graphic coordinate system of XY (with a setting of 0).
- [ZY] soft key
  This soft key selects the graphic coordinate system of ZY (with a setting of 1).
- [YZ] soft key
  This soft key selects the graphic coordinate system of YZ (with a setting of 2).
- [XZ] soft key
  This soft key selects the graphic coordinate system of XZ (with a setting of 3).
- [ZX] soft key
  This soft key selects the graphic coordinate system of ZX (with a setting of 4).
- [XYZ] soft key
  This soft key selects the graphic coordinate system of XYZ (with a setting of 5).
- [YXZ] soft key
  This soft key selects the graphic coordinate system of YXZ (with a setting of 6).
- [YZX] soft key
  This soft key selects the graphic coordinate system of YZX (with a setting of 7).
- [OK] soft key
  This soft key changes the current graphic coordinate system to the graphic coordinate system selected by one of the soft keys above.
- [CANCEL] soft key
  This soft key cancels the graphic coordinate system selected by one of the soft keys above to return to the original graphic coordinate system.

NOTE
The graphic coordinate system selected here is set in the graphic parameter for the graphic coordinate system.

- Rotating the graphic coordinate system

The following soft keys displayed by step 6 are used.

- [↑] soft key
  This soft key rotates the graphic coordinate system upward.
- [↓] soft key
  This soft key rotates the graphic coordinate system downward.
- [←] soft key
  This soft key rotates the graphic coordinate system to left.
- [→] soft key
  This soft key rotates the graphic coordinate system to right.
- [CW] soft key
  This soft key rotates the graphic coordinate system clockwise.

- [CCW] soft key
  This soft key rotates the graphic coordinate system counterclockwise.
- [OK] soft key
  This soft key changes the rotation angle of the current graphic coordinate system to the one set by one of the soft keys above.
- [CANCEL] soft key
  This soft key cancels the rotation of the graphic coordinate system performed by one of the soft keys above to return to the original graphic coordinate system.

NOTE
1 Set the travel increment made by one rotation operation in parameter No. 14716.
2 The rotation angle of the graphic coordinate system set here is not set in the graphic parameter for rotation angle.

- Drawing of a tool

In animation drawing, not only a blank figure but also a tool figure is drawn in three dimensions.

NOTE
The following function is necessary to draw in the tool figure.
- Tool geometry size data 100/300 pairs

Figure data

For tool figure drawing, the following data is used:

- Tool geometry size data 100/300 pairs
- Tool compensation data (tool-nose radius compensation value, virtual tool tip direction, cutter compensation value)
- Parameters

The table below indicates the drawable types of tools and figure data of each type of tool.

Table 13.2.2.2 (a) List of drawable tools (for turning)

Tool name	Tool geometry size data	Tool compensation	Parameter No.	
General-purpose tool	Setting		Tip position	No.27350#0
	Cutting edge angle		Cutting edge length	No.27351
	Tool angle		Holder length	No.27352
			Holder width	No.27353
			Holder length 2	No.27354
			Holder width 2	No.27355
Threading tool	Setting		Tip position	No.27356#0
	Tool angle		Cutting edge width	No.27357
			Holder length	No.27358
			Holder width	No.27359
Grooving tool	Setting		Tip position	No.27360#0
	Tip width		Holder length	No.27361
	Tip length		Holder width	No.27362
Round-nose tool	Setting	Tool-nose radius	Tip position	No.27363#0
	Tip length	Virtual tool tip	Holder length	No.27364
			Holder width	No.27365

Tool name	Tool geometry size data	Tool compensation	Parameter No.	
Point nose straight tool	Setting Cutting edge angle Tool angle		Tip position Cutting edge length Holder length Holder width Holder length 2 Holder width 2	No.27366#0 No.27367 No.27368 No.27369 No.27370 No.27371
Versatile tool	Setting Tip width Tip length		Tip position Holder length Holder width	No.27384#0 No.27385 No.27386

Table 13.2.2.2 (b) List of drawable tools (for milling)

Tool name	Tool geometry size data	Cutter compensation	Parameter No.	
Drill	Setting Tool angle	Diameter	Tip length	No.27372
counter sink tool	Setting Cutter diameter	Small diameter	Tool angle Tip length Cutter length Shank length Shank diameter	No.27375 No.27376 No.27377 No.27378 No.27379
Flat end mill	Setting	Diameter	Tip length	No.27373
Ball end mill	Setting	Diameter	Tip length	No.27380
Tap	Setting	Diameter	Tip length	No.27374
Reamer	Setting	Diameter	Tip length	No.27381
Boring tool	Setting	Diameter	Tip length	No.27382
Face mill	Setting	Diameter	Tip length	No.27383

NOTE
In lathes system, the round-nose tool and milling tools can be drawn only when the cutter compensation/tool-nose radius compensation function is provided.

Timing of drawing

If a code indicated below is specified during drawing execution, a tool figure is drawn according to the tool geometry size data that has the same number as the code.

- With a lathe system
  T code command
  If three numbers, namely, a tool number, tool geometry offset number, and tool wear offset number, are specified in a T code, the tool geometry size data that has the same number as the tool geometry offset number is obtained.

NOTE
When the tool management function is provided, a tool offset is specified with a D code. So, the tool geometry size data that has the offset number indicated by the D code is obtained.

- With a machining center system
  D code command

If the tool geometry size data corresponding to a specified number does not exist or the tool geometry size data is not set correctly, tool drawing is disabled with the warning "ILLEGAL SETTING OF TOOL FIGURE DATA".

13.2.3 Programmable Data Input (G10) for Blank Figure Drawing Parameters

Overview

Each of the drawing parameters for a blank figure, a position, and dimensions that are for the automatic scaling of the drawing areas of tool path drawing and animation drawing with the dynamic graphic display function can be set with an NC program, using the programmable data input command (G10 command).

Format

G10 L90 P_ IP_ I_ J_ K_ ;

P_ : Blank figure
IP_ : Addresses of the basic three axes and the reference position of a blank
I_,J_,K_ : Dimensions of a blank

NOTE
1 This command is enabled only during the execution of drawing. It is ignored during normal automatic operation.
2 This command is a one-shot G code.
3 This command must be specified in a single block.

Explanation

- Blank figure (P_)

Specify the type of a blank figure with either of the following settings for shapes.

Setting	Figure
0	Column or cylinder (parallel with the Z-axis)
1	Rectangular parallelepiped

The specified value is set in parameter No. 11343.

- Reference position of a blank (IP_)

Specify the reference position of a blank with coordinates (X,Y,Z) in the workpiece coordinate system.

NOTE
Specify the reference position of a blank with coordinates (X,Y,Z) in the machine coordinate system in case of bit 3 (BGM) of parameter No.11329=1.

The specified value is set in parameter No. 11344.

- Dimensions of a blank (I_,J_,K_)

For the shape of each blank, specify the dimensions of the blank as follows:

Blank figure	Address I	Address J	Address K
Rectangular parallelepiped	Length in X-axis direction	Length in Y-axis direction	Length in Z-axis direction
Column	Diameter of column	0	Length of column

Blank figure	Address I	Address J	Address K
Cylinder	Diameter of outer circle of cylinder	Diameter of inner circle of cylinder	Length of cylinder

The specified value are set in parameter No.11345 (address I), parameter No.11346 (address J), and parameter No.11347 (address K).

- Relationship between the position and dimensions of a blank

The relationship between the position and dimensions of a blank is shown in the figure below (Fig. 13.2.3 (a)).

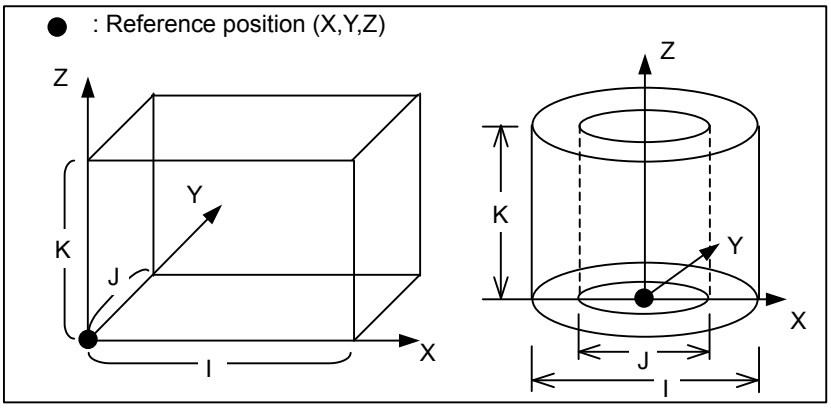


Fig. 13.2.3 (a)

- Operation to be performed when this command is issued

When this command is executed in the background operation, the specified values are set in the drawing parameters for the blank figure, reference position, and dimensions that correspond to the specified arguments, and the drawing area is automatically set again with the values set.
This causes this command and subsequent commands to perform drawing in the new drawing area.

NOTE
Before this command, drawing is performed as described below.
- The tool path is erased.
- In animation drawing, the blank figure is initialized.
- The drawing start position and the tool figure become ineffective.

Thus, issue this command at the start of a drawing program.

13.2.4 Warning Messages

Warning message	Content
START REJECTED	This program cannot be drawn.
NO PROGRAM SELECTED	No drawing target program is selected.
UNAVAILABLE COMMAND IS IN DRAWING	An NC statement/macro statement that can not execute drawing is instructed.
ILLEGAL SETTING OF GRAPHIC PARAMETERS	Graphic parameters are not set correctly.
ILLEGAL SETTING OF TOOL FIGURE DATA	Tool figure data are not set correctly, or tool figure data are not exist.

13.2.5 Note

NOTE

1 The coordinates used in drawing are absolute coordinates.
Therefore, even if the coordinate system is changed while it is drawing, it draws in the coordinate system when having begun to draw.

2 The drawing target axes are the basic three axes.
The basic three axes are the X-axis, Y-axis, and Z-axis on the basic coordinate system set with parameter No.1022.

3 In a drawing program having a sequence of contiguous small blocks, drawing requires longer processing time, possibly causing machining time to become longer than actual machining time.

4 When drawing is executed, data is treated as described below.

(1) Parameters
The same parameters as for automatic operation are used. However, parameters cannot be rewritten with a command such as the G10 command. If an attempt is made to rewrite a parameter with programmable parameter input (G10L52), the command is ignored.
However, by setting bit 7 (GST) of parameter No.11329, drawing can be temporarily stopped with the warning "UNAVAILABLE COMMAND IS IN DRAWING".

(2) Tool compensation value/workpiece origin offset/extended workpiece origin offset/macro variable/fixture offset data, etc.
For each of drawing execution and automatic operation, separate data is used. At the start of drawing, data for drawing is produced by copying data for automatic operation. In subsequent operations, the data for drawing and the data for automatic operation are treated independently of each other. So, even if data is rewritten with a command such as G10, drawing execution and automatic operation do not affect each other. Note, however, that data rewritten in drawing execution is not reflected in the data for automatic operation but is deleted.

(3) Tool life management data, tool number offset data, 3-dimensional error correction data
These data items are not used for drawing execution. So, if an attempt is made to rewrite data with G10, the command is ignored.
However, by setting bit 7 (GST) of parameter No.11329, drawing can be temporarily stopped with the warning "UNAVAILABLE COMMAND IS IN DRAWING".

13.2.6 Restrictions

- Simultaneous drawing with multiple paths

This function does not support simultaneous tool path drawing by program execution with multiple paths. For example, the tool paths of programs executed simultaneously with multiple paths on a 2-path or 3-path lathe or a combined machine tool cannot be drawn. This function supports only drawing executed singly with a single path.

- Simultaneous multi-path display

The screens of this function do not support simultaneous multi-path display based on the setting of parameter No. 13131 and No. 13132.

- Functions that operate differently in drawing execution and automatic operation

The operations of the following functions in drawing execution differ from the operations in automatic operation:

1. Operations that differ, depending on the custom macro
   1) Interface signal
      #1000 to #1035 are assumed to be 0 at all times.
   2) Message output
      A message to be output with #3006 is not displayed but is ignored.
   3) clock
      #3001 and #3002 are ignored. So, note that drawing is not continued, for example, if the following is specified:
      #3001=0 ;
      WHILE [#3001 LE 100] DO1 ;
      END1 ;
   4) Mirror image
      #3007 is assumed to be 0 at all times.
   5) Program-restarted state
      #3008 is assumed to be 0 at all times.
   6) External output command
      BPRNT, DPRNT, POPEN, and PCLOS are ignored.

2 Functions that perform partly different operations
   1) When G28 (automatic reference position return) is specified, up to the intermediate point is drawn.
   2) When G29 (automatic return from the reference position) is specified, drawing is performed from the intermediate point.
   3) When G27 (reference position return check) is specified, no reference position return check is made.
   4) No stored stroke limit check is made.
   5) When G31 (skip function), or G31.1, G31.2, or G31.3 (multistage skip function) is specified, drawing is performed up to the specified position, regardless of the skip signal.
   6) When G60 (single direction positioning) is specified, drawing is performed directly up to the specified position at all times even if the positioning direction is opposite.

3. Functions that perform different operations
   If the following functions are specified, the operations described below result:
   1) G02.2/G03.2 (involute interpolation)
      Circular interpolation is performed.
   2) G02.3/G03.3 (exponential interpolation)
      Linear interpolation is performed only with linear axes.
   3) G07 (virtual axis interpolation)
      Circular interpolation is performed.
   4) G07.1 (cylindrical interpolation)
      Linear interpolation is performed only with linear axes.

- Functions that do not support drawing

The following functions are ignored in drawing execution:

1) G04 (dwell)
2) G20, G21 (inch/metric switch)
3) Auxiliary function (M, S, T, B)
4) G22, G23 (stored stroke limit check on/off)

5) G10.6 (tool retract data setting)
6) G81.1 (chopping)
7) G25/G26 (spindle variation detection on/off)
8) G10 (programmable data input)

NOTE
If G10 (programmable data input) is specified, drawing can be temporarily stopped with the warning "UNAVAILABLE COMMAND IS IN DRAWING" by setting bit 7 (GST) of parameter No.11329.

The following functions are warning in drawing execution:
1) G02.4/G03.4 (3-dimensional circular interpolation)
2) M198 (external subprogram call)

- Functions that support drawing

The operations performed with the following program commands can be drawn:
1) G00 (Positioning)
2) G01 (Linear interpolation)
3) G02/G03 (Circular interpolation)
   However, helical interpolation with the instruction axis other than basic three axes cannot be drawn.
4) G17/G18/G19 (Plane selection)
5) G33/G34 (Threading)
   However, it is drawn as linear interpolation.
6) G35/G36 (Circular threading)
   However, it is drawn as circular interpolation.
7) G40/G41/G42 (Cutter compensation/cancel)
8) G52 (Local coordinate system)
9) G53 (Machine coordinate system selection)
10) G54 to G59 (Workpiece coordinate system selection)
11) G54.1 (Extended workpiece coordinate system selection)
12) G65 (Macro call)
13) G68/G69 (Coordinate system rotation, 3-dimensional coordinate system conversion/cancel)
14) G90/G91 (Absolute/incremental command)
15) G92 (Workpiece coordinate system change)
16) G92.1 (Workpiece coordinate system preset)
17) G94/G95 (Feed per minute/Feed per rotation)
18) G96/G97 (Constant surface speed control/cancel)
19) M98 (Subprogram call)
20) G90/G92/G94 (Canned cycle, only Lathes system)
21) G71/G72/G73/G74/G75/G76/G70 (Multiple repetitive cycle, only Lathes system)
22) G83/G84G85/G87/G87/G88/G80 (Canned cycle for drilling, only Lathes system)
23) G73/G74/G76/G81/G82/G83/G84/G85/G86/G87/G88/G89/G80
   (Canned cycle for drilling, only Machining center system)

NOTE
1 When a lathe system is used, refer to the G codes above as codes of G code system B.
2 It is possible to draw with the G68 (Coordinate system rotation, 3-dimensional coordinate system conversion) instruction only in the tool path drawing.
  And, the display of coordinates when instructing in G68 is a coordinate value on the workpiece coordinate system.
3 In threading cycle of canned cycle and multiple repetitive cycle, movement of thread chamfering is not drawn.

NOTE
4 In animation drawing, shape by the movement of back boring cycle command is different from actual shape.
5 In animation drawing, movement to shift amount at the bottom of a hole in the fine boring cycle and back boring cycle command is not drawn by setting bit 1 (ABC) of parameter No.11349=0.

- Operations that do not support drawing

The following operations cannot be drawn:
1) Backward movement based on the retrace function
2) Forward movement/backward movement/resumed forward movement based on the manual handle retrace function
3) Movement made with skip signal input for a skip command or multistage skip command
4) Movement on an axis based on PMC axis control
5) Movement on an axis based on real-time custom macro specification
6) Operation based on manual interrupt, manual handle interrupt, etc.
7) Operation based on synchronous control, composite control, and superimposed control
8) Operation based on mirror image of facing tool posts
9) Operation based on the 5-axis machining function
10) Operation based on the high-speed, high-precision function (AI contour control I, AI contour control II and high-speed processing)
11) Operation based on nano smoothing or NURBS interpolation
12) Operation based on the program coordinate system switching function
13) Operation based on the offset memory switching function
14) Operation based on Cs contour control
15) Operation based on Spindle positioning.

- Operation of the polar coordinate interpolation command

- Paths that can be drawn in polar coordinate interpolation mode
  In polar coordinate interpolation mode, the paths due to the move commands of the following axes, which make up the polar coordinate interpolation plane, are drawn.

Plane	Linear axis	Rotary axis (hypothetical axis)
X-C	X axis of basic 3 axes	C axis (Center of rotation is X axis of basic 3 axes)

  In the above configuration, set the axis number of the rotation axis to be drawn in parameter No. 14717.
  If a linear axis (parameter No. 5460) and a rotation axis (parameter No. 5461) other than those of the above configuration are set, drawing is not performed correctly.

- Drawing due to the move commands of axes other than those of the polar coordinate interpolation plane in polar coordinate interpolation mode
  In polar coordinate interpolation mode, the paths due to the move commands of the axes making up the above-mentioned polar coordinate interpolation plane and of the axes perpendicular to that plane (those of the basic three axes) are drawn.
  The paths due to the move commands of axes not relating to polar coordinate interpolation cannot be drawn.

- Cancellation of polar coordinate interpolation mode in drawing operation
  The cancellation of polar coordinate interpolation mode in drawing operation is performed with the following:
  - Execution of the G13.1 command in drawing operation
  - Termination of drawing operation

- Origin point of the polar coordinate interpolation plane in drawing
  If bit 3 (BGM) of parameter No. 11329 is 0, the coordinates used for drawing are absolute ones, so that even if the coordinate system is changed with the workpiece coordinate system or local coordinate system command, the origin point of the coordinate system at the time drawing is started is the origin point of the polar coordinate interpolation plane.

- Display of the current position on the drawing screen
  As the current position in polar coordinate interpolation mode, the specified position in the Cartesian coordinate system is displayed.

NOTE
If the polar coordinate interpolation function is enabled and a non-zero axis number is set for the rotation axis (parameter No. 14717), the coordinates of the axis number set for the rotation axis are displayed as the coordinates of the rotation axis.

- Rotation center for a blank figure during animation drawing
  The origin point of the Cartesian coordinate system of the polar coordinate interpolation plane is the origin of the workpiece coordinate system or the local coordinate system at the time polar coordinate interpolation mode (G12.1) is specified. Thus, as the reference point position of the blank to be set as the drawing parameter blank (position), set a position by considering that rotation is performed with the origin point of that coordinate system as the center.
  For example, if a cylindrical shape is set as the drawing parameter blank (figure), set the reference point position of the blank so that the Z-axis of the above-mentioned coordinate system is at the rotation center of the cylinder.

- Drawing on the tool path graphic (tool position) screen
  If the tool position is drawn on the tool path graphic (tool position) screen through automatic operation, the tool position due to the movement of a rotation axis in polar coordinate interpolation mode cannot be drawn.
  In polar coordinate interpolation mode, only movements along linear axes are drawn.

- Dynamic switching of diameter/radius specification in the background operation

It is possible to perform drawing by switching the amount-of-travel setting on each axis that is to be specified in the drawing program to a diameter or radius value, using programmable switching of diameter/radius specification (G10.9 command) of the function of dynamic switching of diameter/radius specification.

When the G10.9 command of the function of dynamic switching of diameter/radius specification is enabled (bit 5 (PGD) of parameter No. 3400 = 1), drawing is performed by internally switching the diameter/radius specification for the move command for each axis in the background operation, as follows:

- If the specified address value is 0 (radius specification) → Drawing is performed with the radius specification.
- If the specified address value is 1 (diameter specification) → Drawing is performed with the diameter specification.

NOTE
1 For the G10.9 command in the background operation, the diameter/radius specification switching status signal of the function of dynamic switching of diameter/radius specification is not output.
2 The diameter/radius specification switching signal of the function of dynamic switching of diameter/radius specification is ineffective to the background operation.
3 The switching due to the G10.9 command in the background operation is canceled if the background operation is stopped.
4 Using this command requires the function of dynamic switching of diameter/radius specification.

- Data items and commands that can be switched
  For the following data item and command, switching of diameter/radius specification is performed according to the switched specification:
  - Move command from a program
  - Current position display

- Data items that cannot be switched
  For the following data items, switching of diameter/radius specification is not performed because they follow the setting of bit 3 (DIAx) of parameter No. 1006.
  - Parameter
  - Offset
  - Workpiece coordinate system

  In custom macros, coordinates that can be read with the system variables indicating position information, listed below.
  #5001 to #5020 and #100001 to #100050: End point position in the immediately preceding block (workpiece coordinate system)
  #5021 to #5040 and #100051 to #100100: Specified current position (machine coordinate system)
  #5041 to #5060 and #100101 and #100150: Specified current position (workpiece coordinate system)
  #5061 to #5080 and #100151 to #100200: Skip position (workpiece coordinate system)

NOTE
For offsets, the settings of bit 1 (ORC) and bit 2 (ODI) of parameter No. 5004 take precedence.

Screen center coordinates and values of blank (position) drawing parameters are handled with radius specification regardless of the setting of parameter bit DIAx.

- Operation of the P-CODE workpiece number search function in the background operation

When drawing is started with the setting of bit 3 (WNS) of parameter No. 11349, the execution macro specified in system variable #8610 is executed before the drawing program.

- Calling conditions and operations
  a) Bit 6 (PWSR) of compilation parameter No. 9002 is set to 1.
  b) The program number of an execution macro is set in system variable #8610 before the start of drawing.

  If the conditions above are met, pressing soft key [START] or [1BLOCK] to execute drawing causes the following operations to be performed:

(1) The execution macro specified with system variable #8610 is called.
If bit 4 (P98) of compilation parameter No. 9163 is 0, the execution macro is called with an operation equivalent to a simple call (G65), and if P98 is 1, it is called with an operation equivalent to a subprogram call (M98).
(2) When the execution macro terminates, the drawing program is executed.

NOTE

1 This function calls an execution macro if the conditions described in a) and b) are met, regardless of the program selected. Thus, if using this function to perform drawing with an execution macro, modify the program of the execution macro so that macro statements, auxiliary functions, and other commands in the execution macro that are not necessary for drawing are not executed, by referring to the system variable below.

System variable	Value	Execution state
#3010	1	Normal state (if drawing is not executed)
	-1	Drawing execution

2 System variable #8610, referenced during the execution of drawing, is the same as that used during automatic operation. If, therefore, the value of the variable is changed to execute drawing, it is necessary to return the variable to the original value before automatic operation is performed.
Also, automatic operation is performed with the variable value at the time drawing is started and, therefore, caution is required so that the value set is not changed during automatic operation.

- Nesting and local variables
  Nesting is accumulated in an execution macro. An execution macro is called with an operation equivalent to a simple call (G65) or a subprogram call (M98) and, therefore, other execution macros can be called from the called execution macro with all calling methods.
  In the case of an operation equivalent to a simple call (G65), the level of local variables is changed to that of the drawing program. That is, the drawing program do not inherit the local variables used in the execution macro.
  In the case of an operation equivalent to a subprogram call (M98), the local variables at the same level as the drawing program are used. That is, the drawing program inherits the local variables used in the execution macro.

- Drawing start position

In tool path drawing, if G92, G52, or G92.1 (machining center system) or G50, G52, or G50.3 (lathe system) is specified at the start of a drawing target program, the position specified with the G code is the drawing start position. If none of these G codes is specified, the end point of the first moving command is the drawing start position.
In animation drawing, the current position when the tool figure is displayed is the drawing start position.

- Use of this function with other functions

When this function is specified, the following functions cannot be used:

- Graphic display
- Manual Guide *i*

- VGA display based on the C Language Executor

If VGA display based on the C Language Executor is used, the VGA window is erased when the screen display is switched to a screen of this function.

So, when using the VGA window, determine the screen of this function by screen number and close the VGA window before switching the screen display.

- Use of the CNC screen display function

The following restriction exists when the screen of dynamic graphic display function is displayed by the CNC screen display function.

1) The drawing screen is not displayed in the CNC screen display function by way of via embedded Ethernet.
   Please use the CNC screen display function by way of the FAST Ethernet board.
2) The drawing screen is not displayed in the CNC screen dual display function.
3) Please do not start / shut down the CNC screen display function when you display the drawing screen. Please start / shut down the CNC screen display function after it switches to other screens.

- Display unit

This function can be used with a 10.4-inch display unit.
On the screens of this function, the vertical soft keys displayed on a 10.4-inch display unit are not displayed.
To use the vertical soft keys, switch the screen display to another screen.

- Use of the Screen hard copy function

If the screen is copied while executing drawing by screen hard copy function, the drawing stops temporarily. The drawing can be restarted by pushing [RESTART] soft key if necessary after the copy operation is finished.

14 VIRTUAL MDI KEY FUNCTION

Chapter 14, "VIRTUAL MDI KEY FUNCTION", consists of the following sections:

14.1 VIRTUAL MDI KEY 1786

14.1 VIRTUAL MDI KEY

Overview

This function is used to perform program editing and changing of various data using the keyboard displayed on the LCD with a touch panel.

Screen on which a CNC screen is displayed in the upper left 1/4 area

Explanation

A CNC screen equivalent to that displayed on 8.4-inch display unit is displayed in the upper left area of the screen, and the keyboard is displayed in the remaining area.

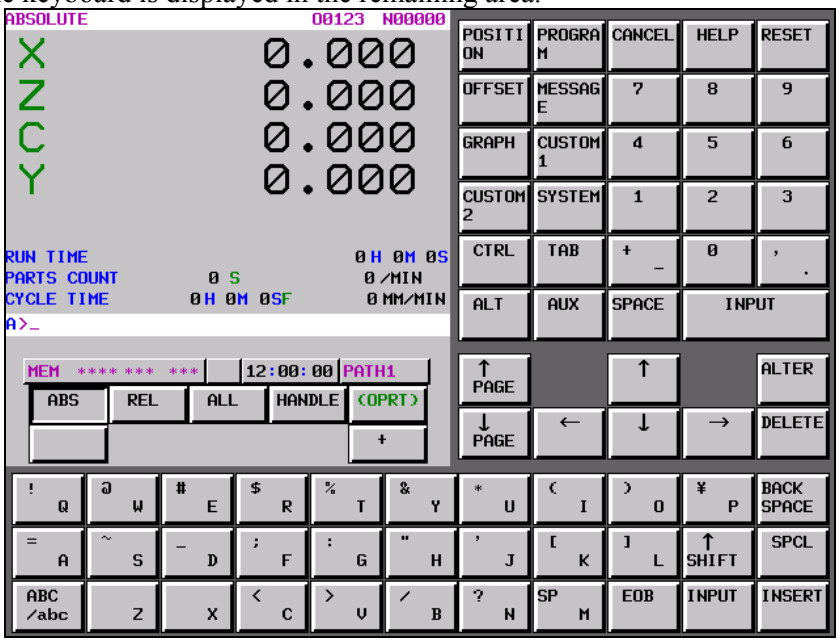


Fig. 14.1 (a) Screen on which a CNC screen is displayed in the upper left area

Operation

- **Input key**

The display "INPUT" on the virtual MDI keyboard is equivalent to the input key.

- **Cancel key**

The displays "BACK SPACE" and "CAN" on the virtual MDI keyboard are equivalent to the cancel key.

- **Shift key**

The display "SHIFT" ↑ on the virtual MDI keyboard is equivalent to the shift key.
Pressing the "SHIFT" key once places the system in the shift state, and pressing the "SHIFT" key again or another key releases the system from the state.
Pressing a key in the shift state causes the character indicated in the upper left corner of the key to be entered.

- Simultaneous pressing of two keys

The operation to be performed for pressing two key simultaneously, such as the "CAN" and "RESET" keys to erase alarm PS100, is as follows:

(1) Press the "SPCL" key. The "SPCL" key is held down, and places the system in SPCL mode.
(2) Press the keys to be pressed simultaneously, one at a time.
(3) Press the "INPUT" key. Of the keys pressed in SPCL mode, the last two are assumed to have been pressed simultaneously. The system is released from SPCL mode, and the SPCL key, which has been held down, returns to the normal state.
Example: "SPCL" → "CAN" → "RESET" → "INPUT"

NOTE
1 In SPCL mode, other keys are disabled until the "SPCL" or "INPUT" key is pressed.
2 Pressing the "SPCL" key in "SPCL" mode causes all keys pressed in SPCL mode to be disabled. The system is released from SPCL mode, and the SPCL key, which has been held down, returns to the normal state.

Screen on which a CNC screen is displayed in the entire area

Explanation

A CNC screen equivalent to that displayed on 8.4-inch display unit is displayed in the entire area. Function keys and the ON/OFF button for virtual keys are displayed at the bottom of the screen.

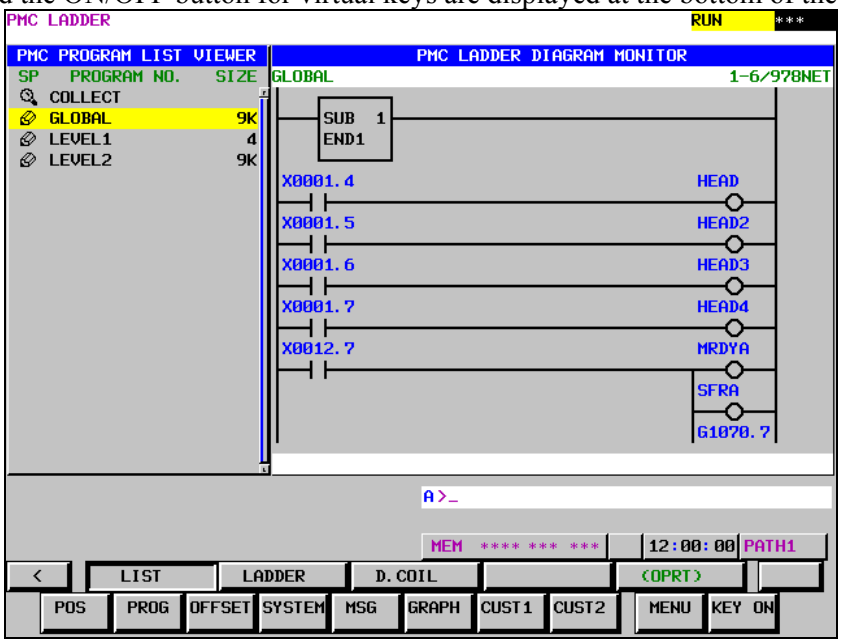
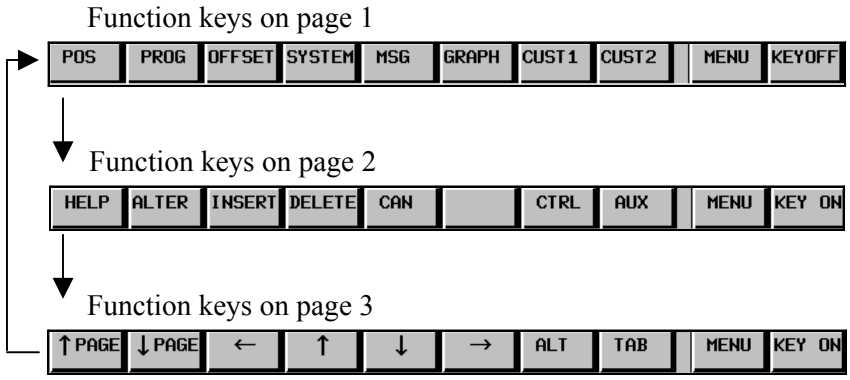


Fig. 14.1 (b) Screen on which a CNC screen is displayed in the entire area

Operation

- Function key page switching

Pressing "MENU" located near the lower right corner of the screen switches the screen to page 1, page 2, page 3, and back to page 1 in this order.



- Display of virtual keys

Pressing "KEY ON" located at the lower right corner of the screen displays virtual MDI keys. The character string on the key top changes to "KEYOFF".
Pressing "KEYOFF" hides virtual MDI keys.

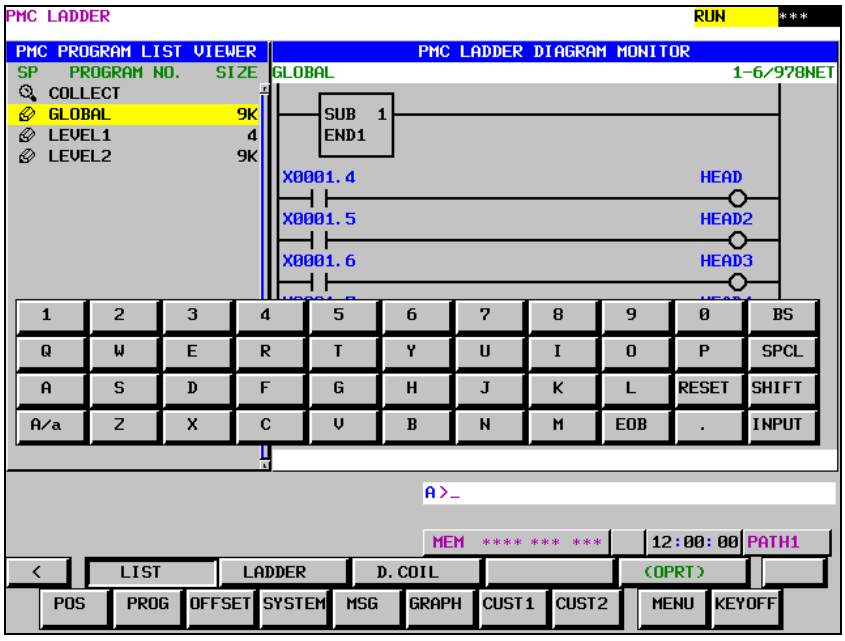


Fig. 14.1 (c) State in which virtual keys are ON

- Input key

The display "INPUT" on the virtual MDI keyboard is equivalent to the input key.

- Cancel key

The displays "BS" and "CAN" on the virtual MDI keyboard are equivalent to the cancel key.

- Shift key

The display "SHIFT" on the virtual MDI keyboard is equivalent to the shift key.
The characters on the key tops change to those that can be entered in the shift state.

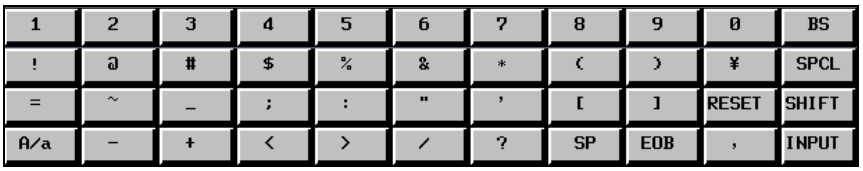


Fig. 14.1 (d) Key tops in the shift state

- Simultaneous pressing of two keys

The operation to be performed for pressing two key simultaneously, such as the "CAN" and "RESET" keys to erase alarm PS100, is as follows:

(1) Press the "SPCL" key. The "SPCL" key is held down, and places the system in SPCL mode.
(2) Press the keys to be pressed simultaneously, one at a time.
(3) Press the "INPUT" key.
Example: "SPCL" → "CAN" → "RESET" → "INPUT"

NOTE
1 In SPCL mode, other keys are disabled until the "SPCL" or "INPUT" key is pressed.
2 Pressing the "SPCL" key in "SPCL" mode causes all keys pressed in SPCL mode to be disabled. The system is released from SPCL mode, and the SPCL key, which has been held down, returns to the normal state.

- Operation history

The operation of virtual MDI key performed by the operator is recorded. The recorded contents are the same as the contents when operating a normal MDI panel.
When the original key "SPCL", "MENU", "KEY ON", "KEY OFF" of virtual MDI key is pressed, the following character string is displayed on the operation history screen and the operation history data (OPRT_HIS.TXT).

Table 14.1 (a) Operation history of original key

Original key	Operation history screen	Operation history data
"SPCL"	{SPCL}	{SPCL}
”MENU”	{MENU}	{MENU}
”KEY ON”, ”KEY OFF”	{KEY}	{KEY ON/OFF}

Moreover, in SPCL mode, not two keys just before pressing "INPUT" key but all operated keys are recorded.
For example)
When the keys of "SPCL", "A", "CAN", "RESET", and "INPUT" are operated in this order, the operation of "A" is not input actually but is recorded.

14.1.1 Limitations

- Display of VGA windows on the C Language Executor

This function uses one VGA window, so that the number of VGA windows that can be used on the C Language Executor is reduced by one.

- Display unit

Display unit on which this function can be used are 10.4 display unit with a touch panel.

15 TEMPLATE PROGRAM FUNCTION

15.1 Template Program Function

Overview

This function is used to manage machining programs, offsets, parameters and so on in folders, as batches. (Such a folder and the data in it are collectively called machining data in the remainder of this manual.)
It can also be used to create templates of machining data in the template folder in advance, so that when new machining data is to be created, it can be created based on the templates. This makes it possible to create similar machining data efficiently.

NOTE
This function requires the 8-level data protection function.

15.1.1 Details

Initial folders

If this function is enabled, two folders are created as initial ones:

- Template folder : SYSTEM/TEMPLATE
- WORKS folder : USER/WORKS

The template folder and the WORKS folder are common to all paths. These folders cannot be deleted.

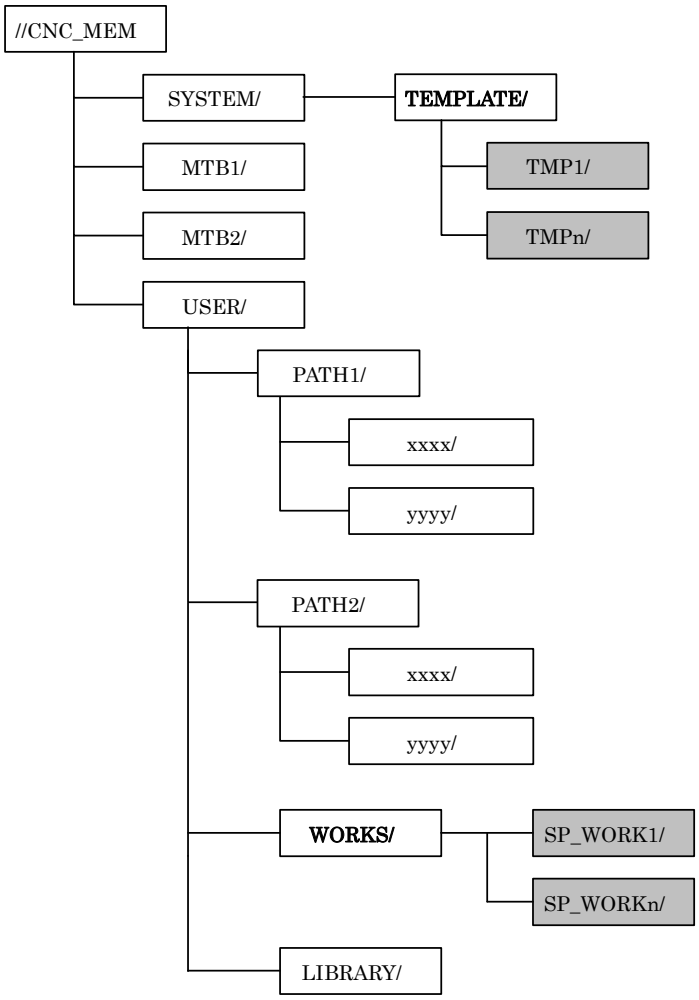


Fig. 15.1.1 (a) Initial folder configuration

NOTE
TMP1, TMPn, SP_WORK1, and SP_WORKn are examples of user-created folders (machining data).

If this function is enabled, but the default foreground and background folders have not been set, it is assumed that the WORKS folder (USER/WORKS) is set.
If this function is enabled, the number of files that the user can register is reduced by 2, as given in Table 15.1.1 (a).

Table 15.1.1 (a)

Program memory capacity	Number of registered programs	Number of registered programs (expansion 1)	Number of registered programs (expansion 2) *2
32Kbyte *1	61	-	-
64Kbyte	61	123	-
128Kbyte	61	248	-
256Kbyte	61	498	-
512Kbyte	61	998	-
1Mbyte	61	998	1998
2Mbyte	61	998	3998
4Mbyte *2	61	998	3998
8Mbyte *2	61	998	3998

*1) FS32*i* only
*2) FS30*i*/31*i* only

Template folder

The template folder is for templates of machining data. Folders created here and the programs in the folders are handled as templates of machining data.
When new machining data is to be created, it is possible to create programs efficiently by creating them based on the templates in the template folder, while avoiding mistakes.
More than one template can be created.

The template folder is protected with the 8-level data protection function.
The template folder is not displayed to those operators whose operation level is lower than 6.

It is not possible to create subfolders in a template.

NOTE
1 Setting a data protection level of the 8-level data protection function to a program in the template folder has no effect. (Objects under the template folder are always regarded to be at level 6.)
2 If the display prohibition attribute is set for a program in the template folder, the program will not be copied when machining data is created.
3 If the operation level is reduced below 6 while a program in the template folder is being subject to background editing, the background editing state can no longer be released. Before changing the operation level, cancel background editing.

WORKS folder

The WORKS folder is used by the user to create machining data.
If an operation for creating a new folder is performed in this folder, machining data will be created based on a template. For details, see “Creating new machining data” in the section on operations.
If a folder is created in a folder other than the WORKS folder, the folder is simply created in the same way as that if the template program function is not used.

Format of machining data

In the machining data folder, main programs and subprograms can be stored. There are no limitations to the program names that can be used, but no subfolders can be created.
Other various data necessary for machining, such as tool offsets, workpiece origin offsets, custom macro variables, and tool management data, can also be stored as programs written in the G10 format. To set each data item, execute a G10 format program, using an M98 subprogram call.

The G10 format is as described below. For details, refer to the related item in the operator's manual.

Setting the workpiece coordinate system

G10 L2 Pp IP_ ;
G11 ;

p=0 : Specify an external workpiece origin offset.
p=1 to 6 : Specify the workpiece origin offset for one of the workpiece coordinate systems 1 to 6.
IP_ : In the case of an absolute command, the workpiece origin offset for each axis. In the case of an incremental command, the value to be added to the workpiece origin offset for each axis that is set. (The addition result is the workpiece origin offset.)

Setting the workpiece origin offset for an additional workpiece coordinate system

```
G10 L20 Pn IP_ ;
G11 ;
```

Pn : Specification code of the workpiece coordinate system for which to set a workpiece origin offset
n : 1 to 48 or 1 to 300
IP_ : Axis address and the setting of the workpiece origin offset

Changing the workpiece coordinate system shift amount

```
G10 P0 IP_;
G11 ;
```

IP : Axis address and the setting of the workpiece coordinate system shift amount

Setting the custom macro variables

```
#500=0.0 ;
#501=0.0 ;
#502=0.0 ;
 :
 :
SETVN500[ABC,DEF]
SETVN501[GHI,JKL]
SETVN502[MNO,PQR]
 :
```

Template format

The format is the same as that of machining data, but has the functions described below.

1. MAIN.TEMPL Main program
   If the name of a main program is set to MAIN.TEMPL, the MAIN part will be renamed to the name specified when machining data is created.

2. ****.TEMPL Program other than a main program
   If the name of a program is set to ****.TEMPL (where **** is any character string), it is prohibited to rename and delete the program under the machining data folder. This prevents the inability to make a subprogram call specified with a template due to the unintentional renaming or deletion by the end user.

Storing data

At present, data that is set with the CNC can be stored as G10 format programs.
Storage folders are folders that are set in the foreground.
The following data can be stored:

- Tool compensation data
- Workpiece coordinate system data
- Workpiece coordinate system shift data
- Macro variable data

NOTE
Workpiece coordinate system shift data can be stored on lathe system machines only.

15.1.2 Operation

Creating a template

A template can be created by only those operators whose operation level is 6 or higher.

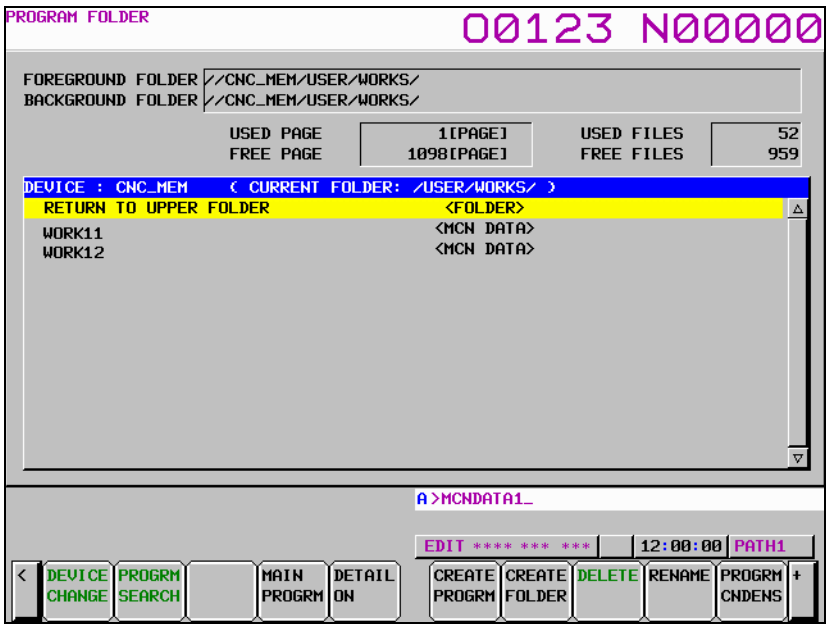
1. Display the program folder screen.
2. Move to the template folder.
3. Enter a folder name and press the soft key [CREATE FOLDER].
4. In the folder created, create a program named MAIN.TEMPL. This will be a main program.
5. Create offset data and so on as G10 format programs. Create the necessary M98 calls in MAIN.TEMPL.

NOTE
1 Subprograms can also be used. Create them, if necessary.
2 For any lines that operators whose operation levels are lower than 6 may not change, insert (R) at the beginning of each of these lines.
3 Another way is to copy machining data and rename the main program to MAIN.TEMPL.

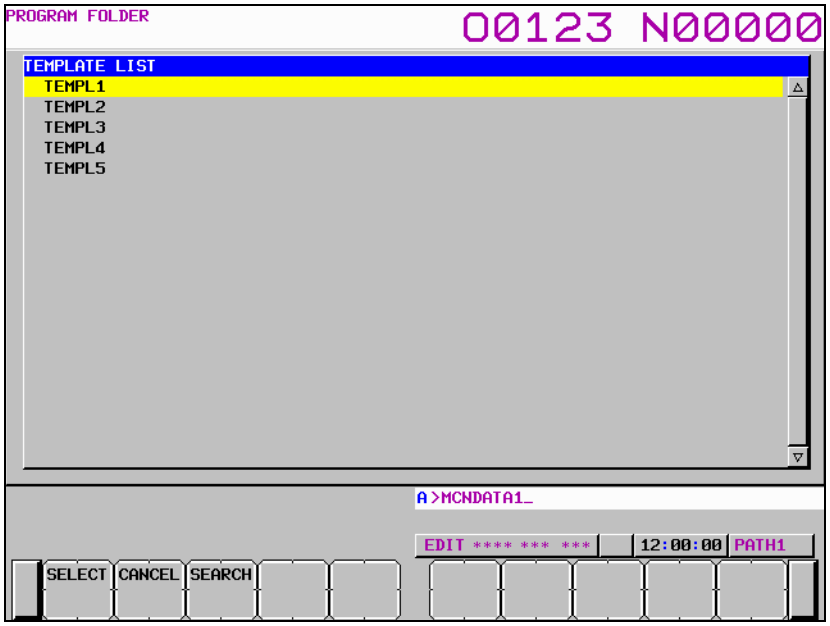
Creating new machining data

If a folder is to be created under the WORKS folder, machining data can be created based on the template folder. If bit 0 (FSB) of parameter No. 11364 is 1, the created folder will be automatically set as a foreground folder and a background folder.

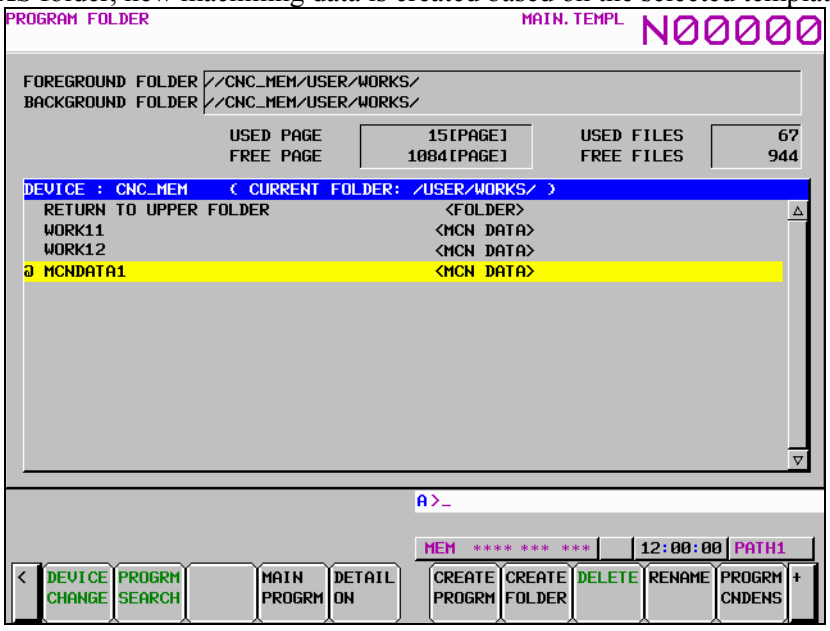
1. Display the program folder screen.
2. Move to the WORKS folder (USER/WORKS).
3. Enter machining data name, and press the soft key [CREATE FOLDER].



4. A list of templates (list of folders in the template folder) is displayed. Using the up and down cursor keys, select a template, and press the soft key [SELECT].
   It is possible to search for a template by entering the name of the template to search for and pressing the soft key [SEARCH].



5. In the WORKS folder, new machining data is created based on the selected template.



NOTE
If an error occurs, such as not enough free space, while machining data is being created, the data being created will be deleted, and the system will return to the state before the start of creation.

Selecting machining data

1. Display the program folder screen.
2. Move to the WORKS folder (USER/WORKS).
3. Move the cursor to the machining data to select.
4. Press soft key [MAIN PROGRM].

When machining data is selected, the program in the folder that is named "machining-data-name.TEMPL" is selected as a program for automatic operation.
The selected machining data folder is set as a foreground folder and a background folder.

NOTE
1 If setting a main program fails, a warning message is displayed. The foreground folder setting and the background folder setting do not change.
2 If setting a foreground folder or a background folder fails, a warning message is displayed.
3 Even if setting a folder fails, the main program of machining data is selected as a program for automatic operation.

Deleting machining data

A machining data file can be deleted from the folder.
1. Display the program folder screen.
2. Move to the WORKS folder (USER/WORKS).
3. Move the cursor to the machining data to delete.
4. Press soft key [DELETE].
5. The message, "DELETE FOLDER?" is displayed. Press the soft key [EXEC].

NOTE
Deletion may fail for reasons such as that an attempt is made to delete machining data selected as a main program and that the machining data contains programs that may not be edited. If this happens, the deletion-in-progress state remains. Cancel the condition that prevents the file from deletion, and delete it again.

Editing machining data

Basically, machining data can be edited in the same way as normal programs. There are, however, some cases in which editing is prohibited. For details, see Subsection, "Protection Function".

Inputting and outputting machining data

It is possible to output and input machining data to and from an external device.

Storing data

Data that is currently set with the CNC can be stored as G10 format programs. Select the data to store, using a soft key.
If the soft key [ALL] is pressed, the data given in Table 15.1.2 (a) is stored as a batch.
The data is stored in the folder that is set in the foreground.

1. Display the program folder screen.
2. If the folder in which to store the data is not a foreground folder, move to the folder in which to store the data and press the soft key [CHANGE FORE].
3. Press soft key [DATA SAVE].
4. Press the soft key of the data to store or the soft key [ALL].



5. Press soft key [EXEC].
   If not saving it, press [CAN].

NOTE
1 Once data storage is executed, no other CNC operations can be performed until the storage is completed. Data storage cannot be interrupted while a file is being stored.
2 The data that can be stored with the soft key [ALL] is the data for the selected path.
3 If the soft key [STOP] is pressed while overwrite storage is being checked with the soft key [ALL], the storage of subsequent data can be interrupted.
4 If the screen is switched to another while data is being stored as a batch, data storage may be interrupted.
5 The soft key [WORK SHIFT] is displayed on lathe system machines only.

Storage file name

The name entered in the key-in buffer before pressing [EXEC] in step 5 of the above procedure will be the storage file name.
Up to 32 characters can be specified as a storage file name.
If no storage file name is specified, a default file name is assigned according to the data.
For details of default file names, see Table 15.1.2 (a), below.
For storage with the soft key [ALL], the storage data will have a default file name.

Table 15.1.2 (a) Default file names

Data	Default file names
Tool compensation data	TOOLOFST
Workpiece coordinate system data	EXT_WKZ
Workpiece coordinate system shift data	WK_SHIFT
Macro variable data	MACRO

If bit 0 (MDP) of parameter No. 138 is 1, a path number is added at the end of a default file name.

Example) For offset data, the file name is as given in Table 15.1.2 (b).

Table 15.1.2 (b) Offset data file names

Selected path number	Bit 0 (MDP) of parameter No. 0138	
	0	1
1	TOOLOFST	TOOLOFST.P-1
2	TOOLOFST	TOOLOFST.P-2

Overwrite saving of data

When bit 1 (COW) of parameter No. 11308 is set to 1, if a file with the same name already exists in the machining data folder, data can be saved.

1. Display the program folder screen.
2. If the folder in which to store the data is not a foreground folder, move to the folder in which to store the data and press the soft key [CHANGE FORE].
3. Press soft key [DATA SAVE].
4. Press the soft key of the data to store or the soft key [ALL].
5. Press soft key [EXEC].
6. If the folder contains a file with the same name, an overwrite conformation message, "*** OVERWRITE?", is displayed (where *** is the file name).
7. To overwrite the file with the file name, press the soft key [REWRITE].
   If not overwrite it, press the soft key [CAN].

NOTE
1 If data is stored as a batch, pressing the soft key [CAN] cancels the present output operation, and continues the storage of the next file.
2 If data is stored as a batch, pressing the soft key [STOP] interrupts the storage of the data subsequent to the data that has been stored.

15.1.3 Protection Function

Protecting the template folder

The template folder is protected with the 8-level data protection function.
The template folder is not displayed to those operators whose operation level is lower than 6. Those operators cannot output or input the template folder.

NOTE
Setting a data protection level of the 8-level data protection function to a program in the template folder has no effect. (Objects under the template folder are always regarded to be at level 6.)

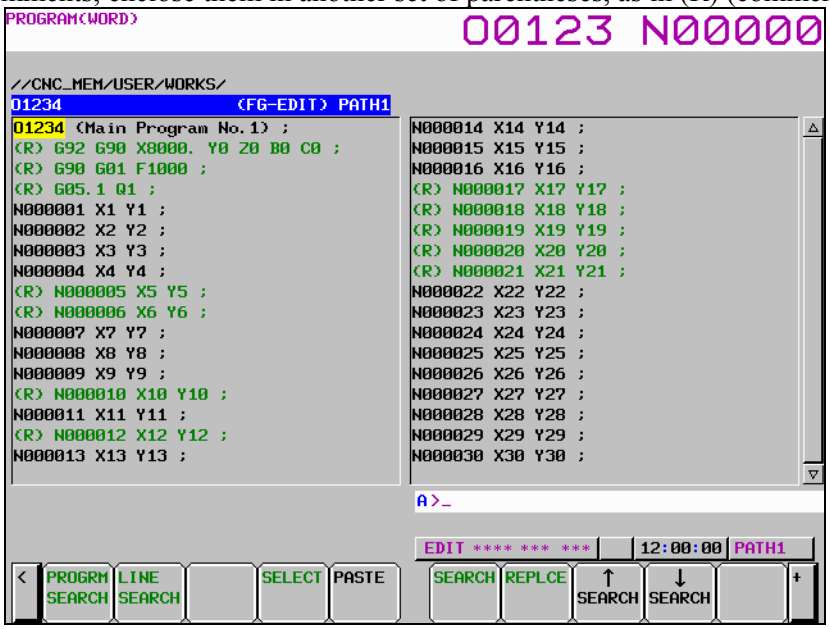
Protecting template programs (Protection in program units)

Those operators whose operation level is lower than 6 cannot perform the following operations on those programs that have program names ending with the six characters ".TEMPL".

- New create
- Delete
- Rename
- Copy
- Move

Protecting template programs (Protection in line units)

By adding (R) at the beginning of a block in a template in advance, it is possible to prohibit those operators whose operation level is lower than 6 from editing machining data on a line-by-line basis.
If adding other comments, enclose them in another set of parentheses, as in (R) (comment data).



- It is not possible to make insertions into, make changes to, and make deletions from a block that has (R) at the beginning.
- It is not possible to make insertions, changes, deletions, and replacements that will cause (R) to appear at the beginning of a block as a result of editing.
- It is not possible to copy, cut, and paste a character string containing (R).
- It is not possible to insert a program containing (R).

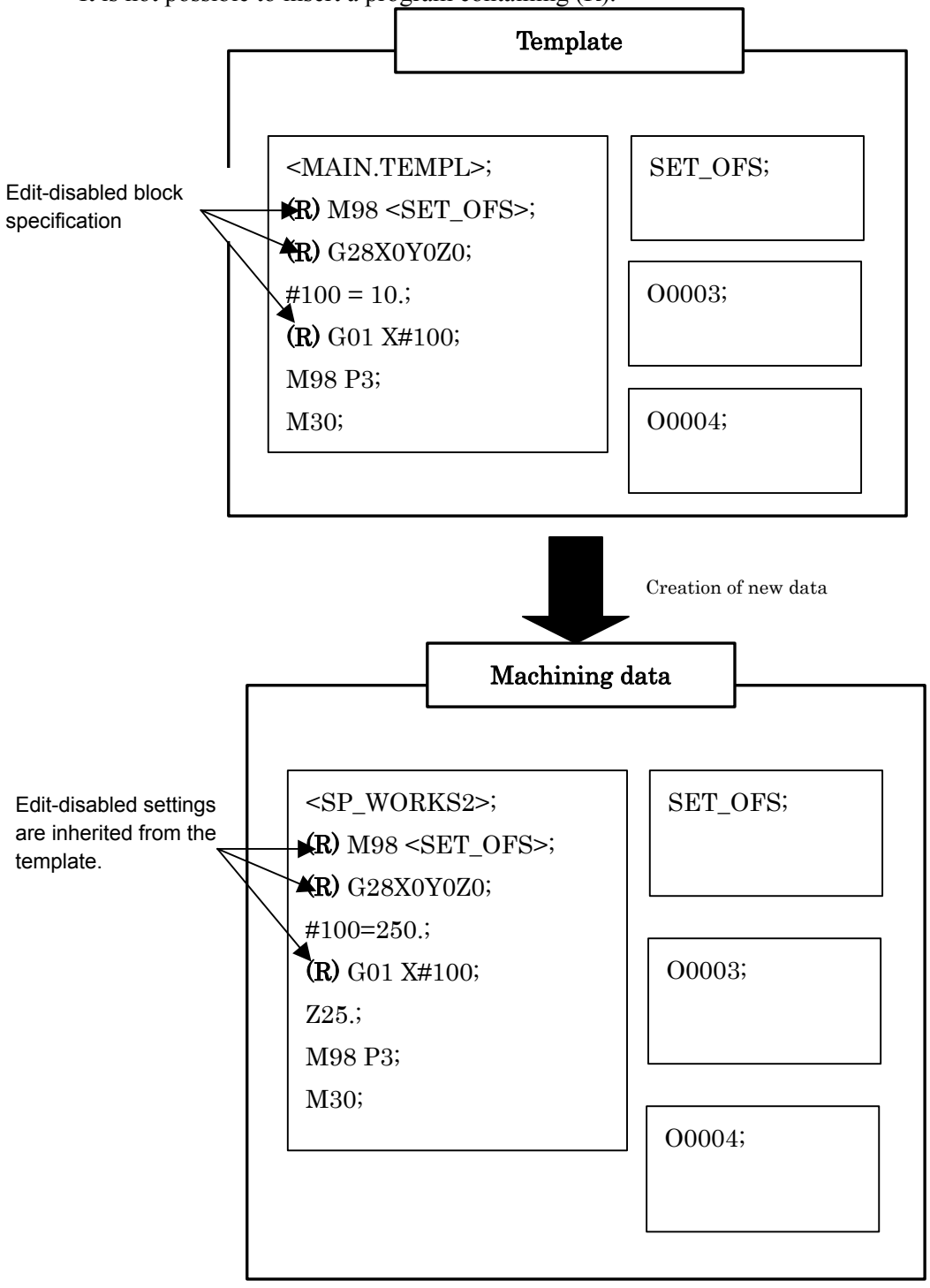


Fig. 15.1.3 (a) Protection of machining data in line units

15.1.4 Limitations

- This function cannot be used together with the limitations of folder manipulation due to bit 6 (FPF) of parameter No. 11302 and bit 7 (CFP) of parameter No. 11304. The settings of FPF and CFP of these parameters are not effective.

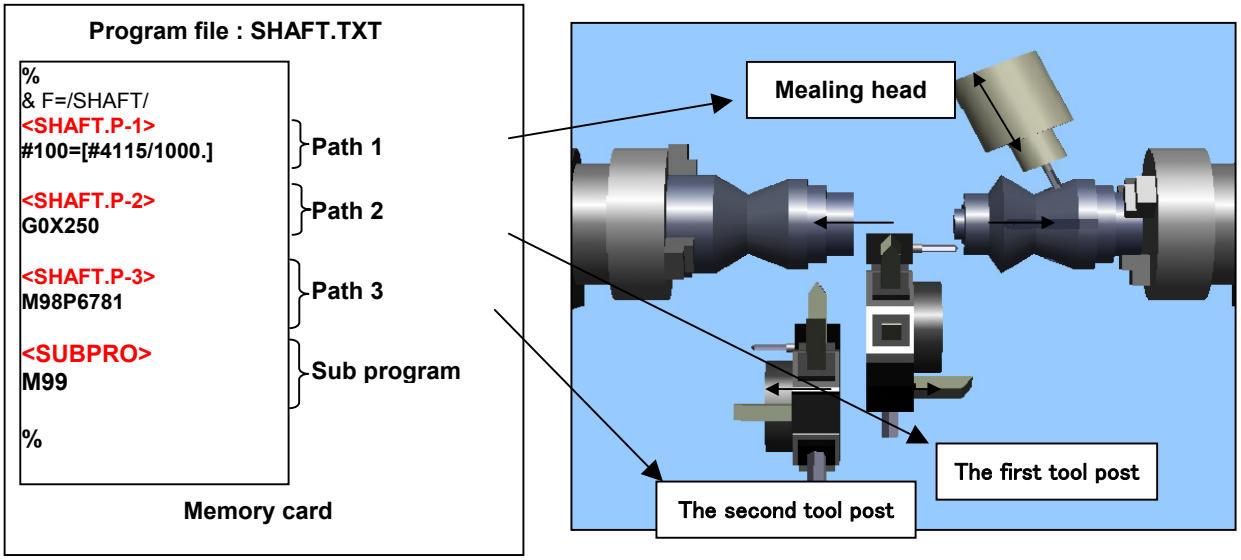
When this function is enabled/disabled, file clearing is performed. Create backups of programs before switching the function between enabled and disabled states.

16 MULTI-PATH PROGRAM MANAGEMENT FUNCTION

16.1 MULTI-PATH PROGRAM MANAGEMENT FUNCTION

Overview

The multi-path lathe and the complex machine that have plural turret or head machine work piece by two or more machining programs.
In this function, multi-path programs, offset data, and parameters, etc. for one part machining are managed together by a folder. Moreover, this improvement easily does making, the selection, the edit, and the I/O of the programs.



Example of complex machining center （One milling head and two tool posts）

NOTE
1 This function is an optional.
2 This function requires the Multi-path editing function option.
3 This function cannot be used with the Template program function at the same time.

16.1.1 Details

Programs for each path, which are necessary for one machining, can be edited simultaneously. Selection, I/O, and renaming of these programs are possible. These operations can be done to the programs in the Multi-path program folder.
Please refer to the following "Individual program input and editing" when the program is handled individually.

16.1.1.1 Multi-path program folder

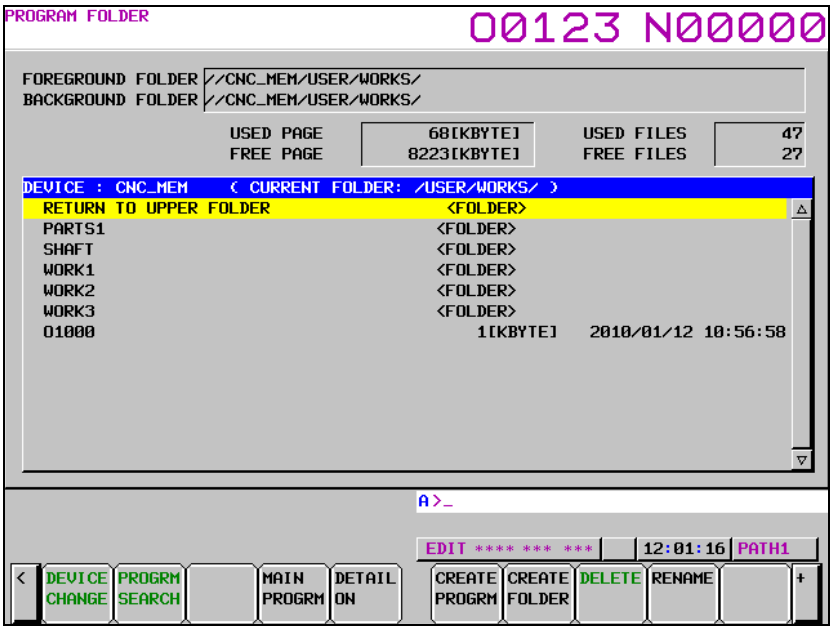
The folder, in which the main programs and the subprograms of each path necessary for the machining, is called the "Multi-path program folder".
Moreover, various data necessary for machining such as the tool offset, the work offset, the custom macro variable, and the tool management data, etc. can be stored in Multi-path program folder as a program of G10 format.

16.1.1.2 Making of machining programs

The program, for each path can be created by the following operations.

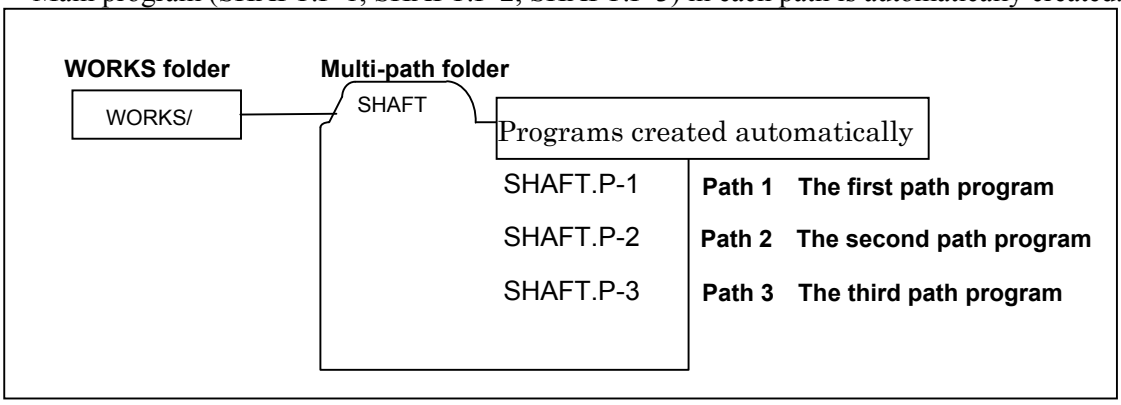
1 Select “all paths EDIT mode”.
2 Press function key 

.
3 Press soft key [FOLDER] to display the program folder screen.
4 Select the WORKS folder (//CNC_MEM/USER/WORKS/).
5 Enter Multi-path program folder name, and press the soft key [CREATE FOLDER].



6 New Multi-path program folder is created in the WORKS folder. The main program in each path is automatically created in the Multi-path program folder at the same time.

Example) When the Multi-path program folder named SHAFT (Three paths system) is created.
Main program (SHAFT.P-1, SHAFT.P-2, SHAFT.P-3) in each path is automatically created.



NOTE
1 The Multi-path program folder can be created only in the WORKS folder.
2 The subfolder cannot be created in the Multi-path program folder.
3 If an alarm such as not enough free space of memory occurs, while Multi-path program folder is being created, the data being created will be deleted, and the system will return to the state before the start of creation.
4 When the program of the path need not be created automatically, set the parameter bit 3(MPC) of parameter No.11352.

WORKS folder (//CNC_MEM/USER/WORKS/)

The Multi-path program folder can be created only in the WORKS folder (//CNC_MEM/USER/WORKS/). When this function is effective, the WORKS folder is automatically created.
When a new folder is created in WORKS folder, this folder is created as the Multi-path program folder. And, the main programs of all paths are created in that folder simultaneously. The name of the programs that are created by the batch making are named "name of Multi-path program folder. P-n". (n is path number)

NOTE
1 WORKS folder cannot be deleted, and cannot be renamed.
2 When this function is effective and the foreground or background default folder is not set, WORKS folder is assumed as a default folder.

Program number

When this function is effective, the number of files that can be registered decreases by one.

Part program storage size	Number of register-able programs	Number of register-able programs expansion 1	Number of register-able programs expansion 2 *2
32Kbyte *1	62	-	-
64Kbyte	62	124	-
128Kbyte	62	249	-
256Kbyte	62	499	-
512Kbyte	62	999	-
1Mbyte	62	999	1999
2Mbyte	62	999	3999
4Mbyte *2	62	999	3999
8Mbyte *2	62	999	3999

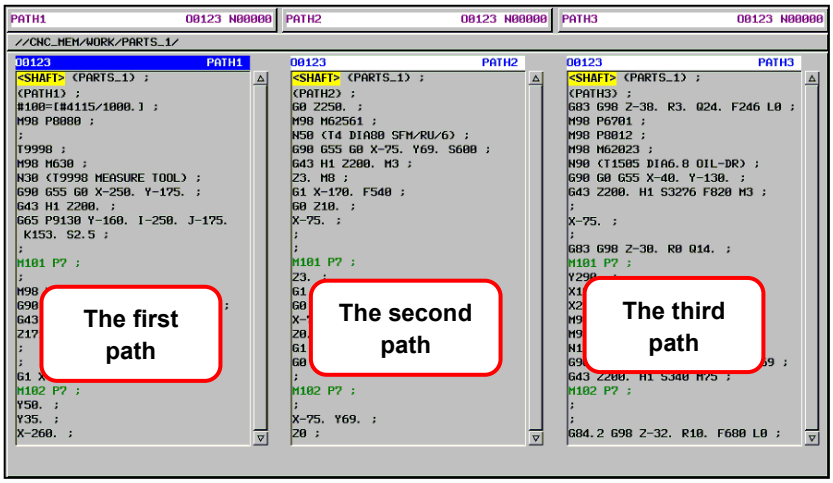
*1) For FS32i only
*2) For FS30i/31i only

16.1.1.3 Edit of multi-path program

By following procedures, main programs can be displayed and edited at the same time by arranging on one screen. Programs of up to three paths can be displayed simultaneously.

1 Perform "Making of machining programs" or "Selection of machining program".
2 Select “all paths EDIT mode“.
3 Press function key PROG.
4 Press soft key [program] to display the program editing screen and edit the program.

Please refer to the chapter of "MULTI-PATH EDITING FUNCTION" of OPERATOR'S MANUAL (Common to Lathe System / Machining Center System) for a detailed operating instruction.
When programs other than the main program such as subprograms or offset data program are edited, refer to the chapter "Individual program input and editing".



Program editing screen

NOTE
It is necessary to set the parameter of multi path simultaneous display function and simultaneous multi path program editing function to the batch edit. For the details, refer to "MULTI PATH DISPLAY AND EDIT" of CONNECTION MANUAL (FUNCTION).

16.1.1.4 Selection of machining program

The main program of each path can be selected by the following operations simultaneously.

1 Select “all paths MEM mode”.
2 Press function key PROG.
3 Press soft key [FOLDER] to display the program folder screen.
4 Select the WORKS folder (//CNC_MEM/USER/WORKS/).
5 Select the Multi-path program folder by cursor key.
6 Press the soft key [MAIN PROGRAM].

By above operation, the main program of each path is selected simultaneously. By pushing the cycle start the machining is started.

NOTE

1 If the main program does not exist even in one path, warning message "MAIN PROG OF NUM OF PATH IS DEFICIENT" is displayed, and it is not possible to select the Multi-path program. If there is a path not used for the machining, please prepare the dummy program or set the bit 3 (MPC) of parameter No. 11352 for the path. As for the dummy program, fill M02; or M30; as follows.
（Example of dummy program ）
O0123;
M30;
%

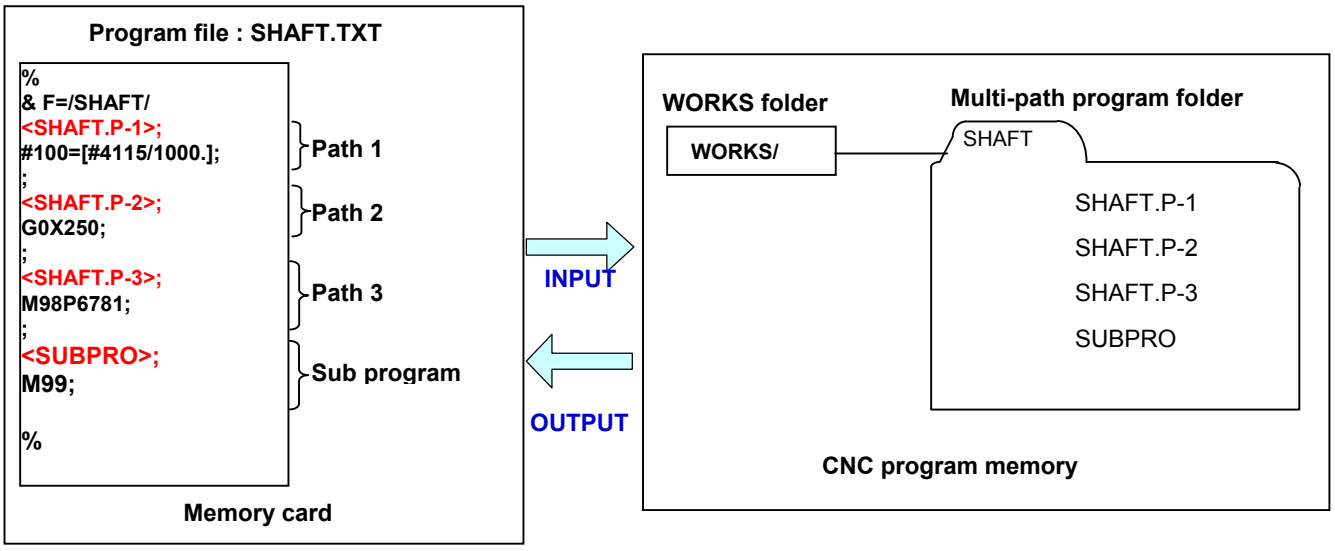
2 The main programs of the path whose bit 3 (MPC) of parameter No. 11352 is set '1' is not automatically changed when the Multi-path folder is selected.

3 In the Multi-path program folder, it is not possible to create, change or search the main program on the edit screen or the program check screen. It is necessary to invalidate the setting of "CAN NOT ENTER MULTIPATH PROG FOLDER" on the operation confirmation function setting screen. Please refer to the chapter "Individual program input and editing" for further details.

16.1.1.5 Input/output of Multi-path program

The main program for each path for a machining can be output to external I/O device (memory card etc.) simultaneously. And, the Multi-path program folder and the programs in it can be input from one program file simultaneously.

Example) Inputting or outputting of multi-path program folder SHAFT (input or output as one file)



Output operation

1 Select "all paths EDIT mode".

2 Press function key 

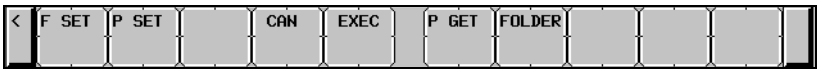
.

3 Press soft key [FOLDER] to display the program folder screen.

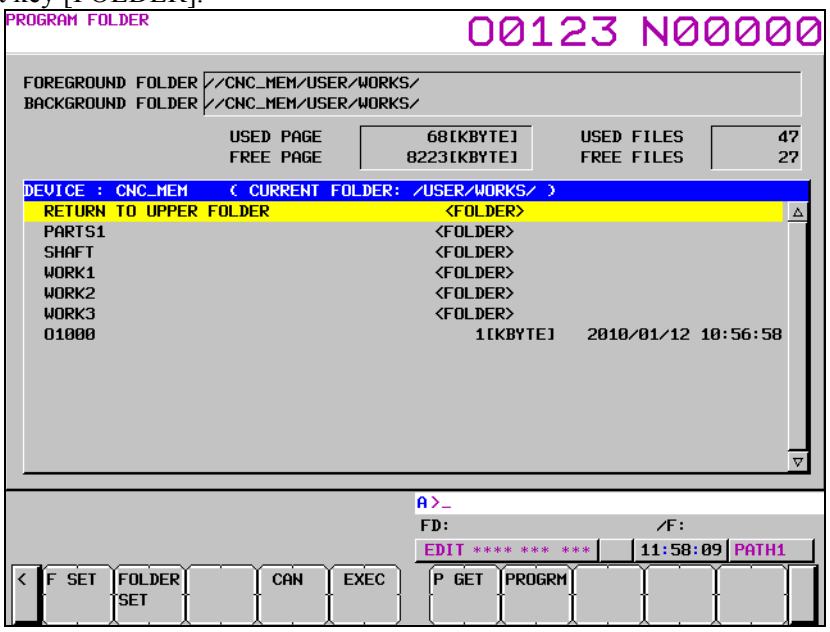
4 Select the WORKS folder (//CNC_MEM/USER/WORKS/).

5 Select the Multi-path program folder by cursor key.

6 Press the soft key [PUNCH]. The following softkey is displayed.

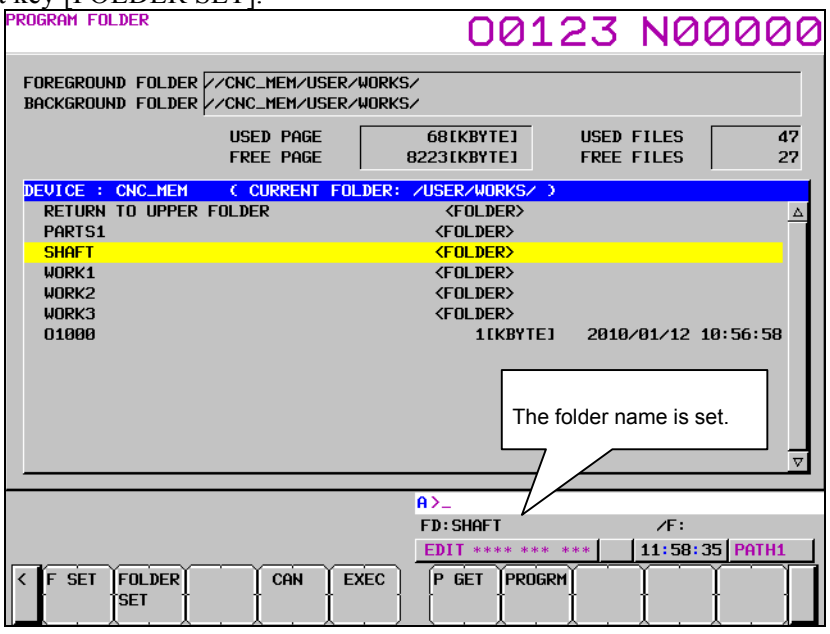


7 Press the soft key [FOLDER].



8 Press the soft key [P GET] or type the name of Multi-path program folder.

9 Press the soft key [FOLDER SET].



10 Press the soft key [P GET] or type the output file name.

11 Press the soft key [F SET].
When the output file name is not specified, 'PARTFLDR.TXT' is assumed as an output file name automatically.

12 Press the soft key [EXEC], and output is started.

Input operation

1 Select “all paths EDIT mode”.

2 Press function key 

.

3 Press soft key [FOLDER] to display the program folder screen.

4 Select the WORKS folder (//CNC_MEM/USER/WORKS/).
5 Press the soft key [READ].
6 Type the input file name, and press the soft key [F SET].
When the input file name is not specified, 'PARTFLDR.TXT' is assumed as becomes an output file name automatically.
7 Press the soft key [EXEC].
The file is input and the Multi-path program folder is created automatically, and then programs of each path are input.

Format of I/O

The format of I/O is the following form.

Example) Outputting Multi-path program folder SHAFT (3 paths system)

```
%
& F=//CNC_MEM/USER/WORKS/        WORKS folder
&F=/SHAFT/                       Multi-path program folder
<SHAFT.P-1>
  :
M30
<SHAFT.P-2>
  :
M30
<SHAFT.P-3>
  :
M30
<SUBPRO>
  :
M99
%
```

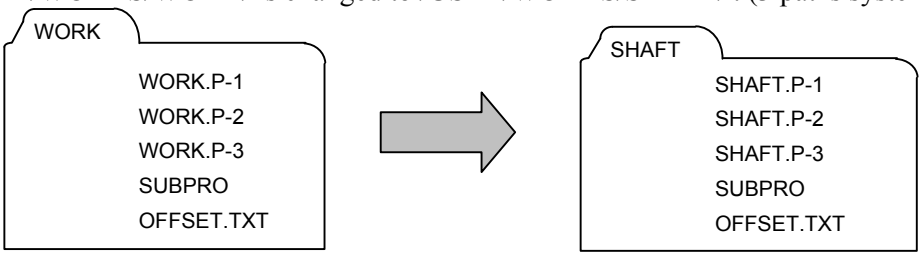
NOTE
1 The bit 0 (MDP) of parameter No. 138 that adds the path number to the extension of the output file is invalid.
2 The bit 7 (FLI) of parameter No. 11364 that select “the FG/BG folder is targeted” or “current folder is targeted” is invalid in the WORKS folder. Input and output operation is performed for a current folder.

16.1.1.6 Name change of the multi-path program folder

The name of Multi-path program folder and main programs in the folder can be changed simultaneously.

1 Select “all paths EDIT mode”.
2 Press function key .
3 Press soft key [FOLDER] to display the program folder screen.
4 Select the WORKS folder (//CNC_MEM/USER/WORKS/).
5 Select the Multi-path program folder by cursor key.
6 Type the changed name, and press the soft key [RENAME].
7 The folder name and the name of the main program in the folder are changed.

Example) /USER/WORKS/WORK/ is changed to /USER/WORKS/SHAFT/ . (3 paths system)



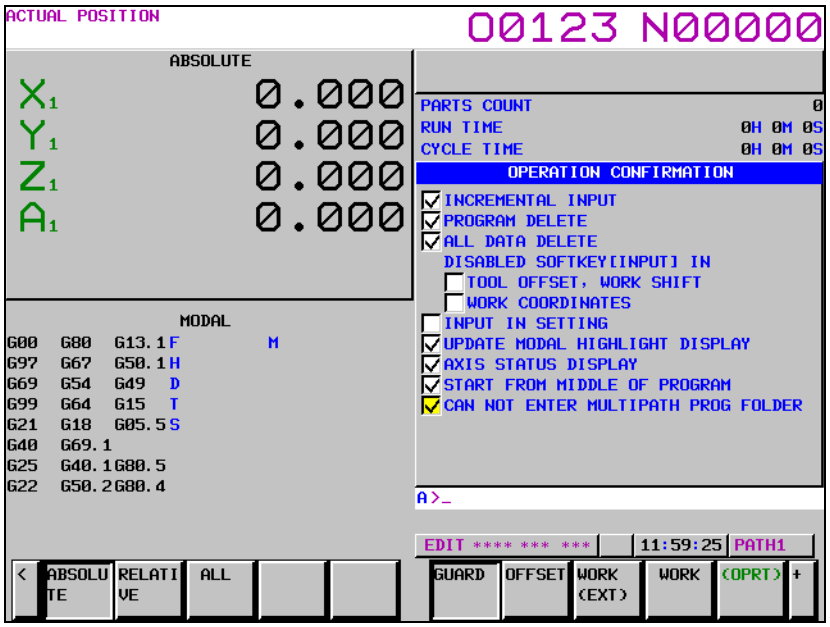
NOTE
The names of the main programs under the selection cannot be renamed.

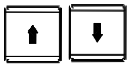
16.1.2 Operation of inputting and editing for each program

The inputting, outputting and editing of sub program and the offset data, etc. can be performed by the following method.

Procedure

1 Select MDI mode.
2 Press function key .
3 Press the continuous menu key  several times until the [GUARD] is appeared.
4 Press the [GUARD] soft key. The setting screen that was displayed the last and belongs to the operation confirmation function and was displayed.(the operation confirmation function setting screen is the first such screen that appears after the system is restarted).
5 If any screen other than the operation confirmation function setting screen is displayed, press the [GUARD] soft key. The operation confirmation function setting screen is displayed



6 Move the cursor to the check box of "CAN NOT ENTER MULTIPATH PROG FOLDER", by pressing the  key.

7 Press the soft key [(OPRT)].

8 Press the soft key [OFF:0].
By this operation it becomes possible to select the Multi-path program folder on the program folder screen. And the individual operation for each multi-path program is permitted.

9 Press function key 

.

10 Press soft key [FOLDER] to display the program folder screen.

11 Select EDIT mode.

12 Move to the folder where the program to be edited is placed. Perform the operating of Edit or inputting and outputting the program.

13 After editing ends move to the WORKS folder (//CNC_MEM/USER/WORKS/) in the program folder screen.

14 Operate ‘1’ to ‘7’, and move the cursor to the check box of "CAN NOT ENTER MULTIPATH PROG FOLDER".

15 Press the soft key [ON:1]. Then, an individual operation of multi-path program is prohibited.

"CAN NOT ENTER MULTIPATH PROG FOLDER"

When this item is effective, it is not possible to move in the Multi-path program folder. Therefore, individual selection, individual input/output, and individual deletion are impossible. Please remove the check on this item when an individual operation is required for each file in the Multi-path program folder in case editing the subprogram.

NOTE
1 When the check mark is put in this item while displaying the folder in the Multi-path program folder on the program folder screen, the protection function doesn't become effective for the folder under the display. Please put the check mark in this item after the cursor is moved out of the Multi-path program folder.
2 Please do not put the check mark in this item when the Multi-path program folder is set to the foreground folder. This setting doesn't become effective to the Multi-path program folder set to the foreground folder after the next power-on. Please put the check mark in this item after changing the foreground folder other than the Multi-path program folder.
3 When the Protection of Data at Eight Levels function is effective, the protection level that can change the parameter data should be selected.

16.1.3 Limitations

When the Multi-path program management function is effective, even if the following parameters are set to 1, those parameters are invalid.

	#7	#6	#5	#4	#3	#2	#1	#0
11302		FPF						

[Input type] Parameter input
[Data type] Bit

#6 FPF Folders that can be used by program management are:
0: Not limited.
1: Limited to other than the path folder corresponding to a selected path.

	#7	#6	#5	#4	#3	#2	#1	#0
11304	CFP							

[Input type] Parameter input
[Data type] Bit

#7 CFP Folders that can be used by program management are:
0: Not limited.
1: Limited to the path folder corresponding to a selected path.

IV. MAINTENANCE

1 ROUTINE MAINTENANCE

This chapter describes routine maintenance work that the operator can perform when using the CNC.

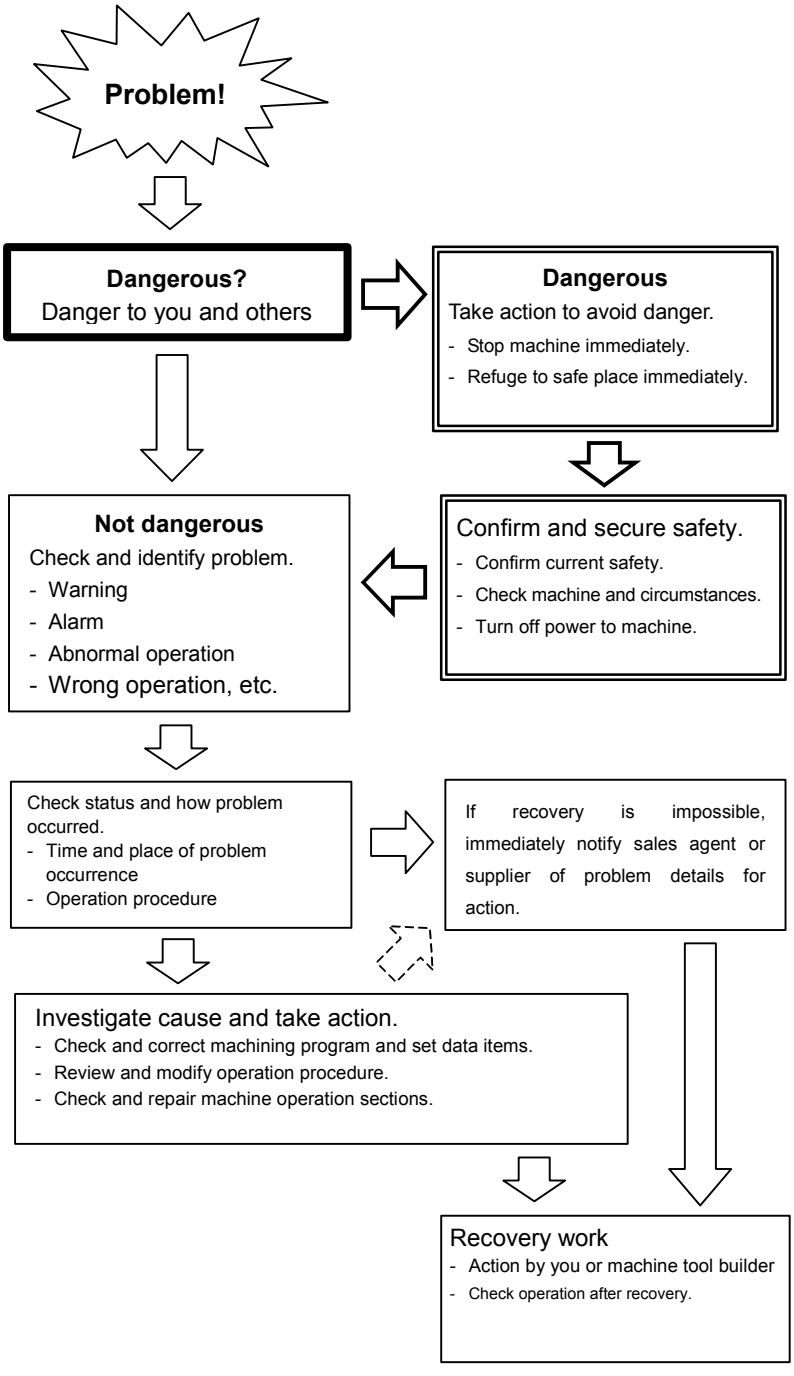
> ⚠ **WARNING**
> Only those persons who have been educated for maintenance and safety may perform maintenance work not described in this chapter.

Chapter 1, "ROUTINE MAINTENANCE", consists of the following sections:

1.1 ACTION TO BE TAKEN WHEN A PROBLEM OCCURRED 1814
1.2 BACKING UP VARIOUS DATA ITEMS.....1815
1.3 METHOD OF REPLACING BATTERY.....1817

1.1 ACTION TO BE TAKEN WHEN A PROBLEM OCCURRED

If an unexpected operation occurs or an alarm or warning is output when the CNC and machine are used, the problem needs to be solved quickly. For this purpose, the status of the problem must be identified correctly, and a proper action must be taken.
The procedure for taking an action for a problem is shown below.



For details of investigation and action on problems arising from the CNC, refer to "TROUBLESHOOTING PROCEDURE" in the Maintenance Manual (B-64485EN) issued by FANUC.

1.2 BACKING UP VARIOUS DATA ITEMS

With the CNC, various data items such as offset data and system parameters are stored in the SRAM of the control unit and are protected by a backup battery. However, an accident can erase the data. By storing the data at another location (outside the CNC), the data, when lost, can be restored.
So, when the machine is started up or data is updated, for example, the data should be backed up (stored outside the CNC).

- Data backup operation

The data items listed below should be backed up. For the method of data output operation, see the chapter of "DATA INPUT/OUTPUT" in this manual.

<1> System parameters
→ See Subsection, “Inputting and Outputting Parameters”.
<2> Machining programs
→ See Subsection, “Inputting and Outputting a Program”.
<3> Tool offset data
→ See Subsection, “Inputting and Outputting Offset Data”.
<4> PMC data
→ See PMC PROGRAMMING MANUAL (B-64513EN).
<5> Pitch error compensation data (when the pitch error compensation function is selected.)
→ See Subsection, “Inputting and Outputting Pitch Error Compensation Data”.
<6> Three-dimensional error compensation (when the three-dimensional error compensation function is selected.)
→ See Subsection, “Inputting and Outputting 3-dimensional Error Compensation Data”.
<7> Custom macro variables (when the custom macro function is selected.)
→ See Subsection, “Inputting and Outputting Custom Macro Common Variables”.
<8> Workpiece coordinate system setting data (when the workpiece coordinate system function is selected.)
→ See Subsection, “Inputting and Outputting Workpiece Coordinates System Data”.
<9> Tool management data (when the tool management function is selected.)
→ See Subsection, “Inputting and Outputting Tool Management Data”.

It is recommended that recording media (such as floppy disks and memory cards) daily used with the machine be used to store data. Stored data should be managed properly so that the data can be restored quickly if a problem occurs.

- Data restoration work

In order to restore lost data to the state of the stored data, input the data backed up according to the previous item into the CNC. For the method of data input operation, see the chapter of "DATA INPUT/OUTPUT" in this manual.

⚠ WARNING
After inputting stored data, do not start an operation immediately. Instead, check that the data is input correctly and that settings are made to meet a desired operation.
If an operation is executed without making this check, the machine and workpiece can be damaged and personal injury can occur due to an unexpected machine movement. Use sufficient care.

⚠ CAUTION

Before recovery of the following data items, consult with the machine tool builder of the machine used:

- System parameters
- PMC data
- Macro programs and custom macro variables
- Pitch error compensation values

NOTE

The method of recovery described in this section is intended just to restore the state of the backed up data, and does not guarantee recovery of the state that was present when the data was lost.

1.3 METHOD OF REPLACING BATTERY

This chapter describes how to replace the control unit backup battery and absolute Pulsecoder battery. This section consists of the following subsections:

1.3.1 Replacing Battery for Control Unit....1818
1.3.2 Battery in the PANEL *i* (3 VDC)....1821
1.3.3 Replacing Battery for Absolute Pulsecoders....1822

Battery for memory backup

Such as Offset data and system parameters are stored in SRAM in the control unit. The power to the SRAM is backed up by a lithium battery mounted on the front panel of the control unit. Therefore, the above data is not lost even if the main power is turned off.
When the battery voltage falls, alarm message "BAT" blinks on the LCD screen and the battery alarm signal is output to the PMC. When this alarm is displayed, replace the battery as soon as possible. In general, the battery can be replaced within one or two weeks of the alarm first being issued. This, however, depends on the system configuration.
If the battery voltage subsequently drops further, backup of memory can no longer be provided. Turning on the power to the control unit in this state causes system alarm to be issued because the contents of memory are lost. Replace the battery, clear the entire memory, then reenter the data.
For this reason, FANUC recommends that the battery be replaced once per year regardless of whether a battery alarm is issued.
The type of battery to be replaced differs depending on the control unit. In some control units, a dedicated lithium battery to be purchased from FANUC is used. In other control units, alkaline dry cells (size D) are used in an external battery case.

NOTE
A lithium battery is installed as standard at the factory. This battery can provide backup for the memory contents for about a year.
When discarding a battery, observe the applicable ordinances or other rules of your local government. Also, cover the terminals of the battery with vinyl tape or the like to prevent a short-circuit.

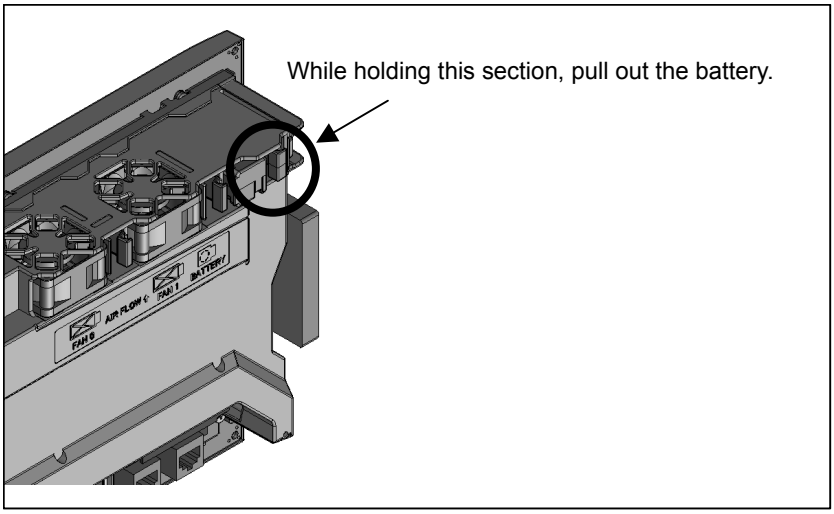
1.3.1 Replacing Battery for Control Unit

When using a lithium battery (for LCD-mounted type control unit)

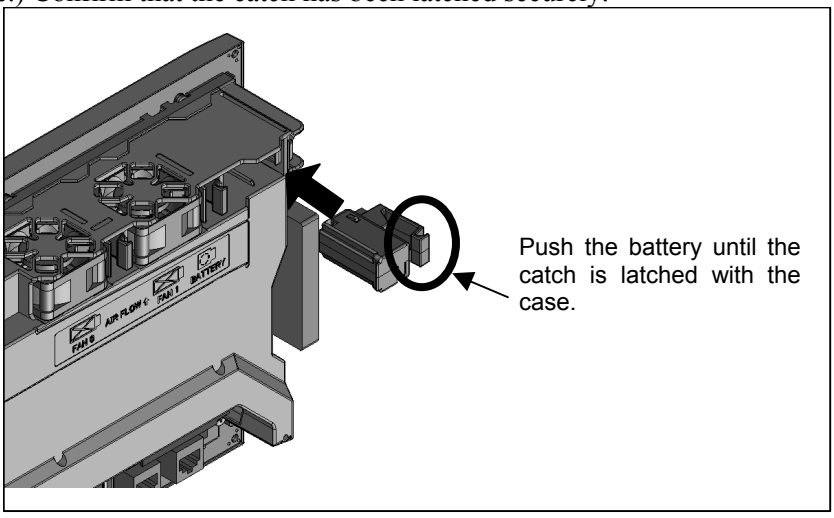
Prepare a new lithium battery (ordering code: A02B-0323-K102).

<1> Turn the power to the machine (control unit) on. After about 30 seconds, turn the power off.

<2> Pull out the lithium battery on the back of the control unit. (While holding the latch of the lithium battery to remove the catch in the case, pull the battery up and out.)



<3> Mount a new lithium battery you get ready beforehand. (Push the battery until the catch is latched with the case.) Confirm that the catch has been latched securely.



⚠WARNING

Using other than the recommended lithium battery may result in the battery exploding. Replace the battery only with the specified lithium battery (A02B-0323-K102).

> ⚠ **CAUTION**
> Steps <1> to <3> should be completed within 30 minutes.
> Do not leave the control unit without a battery for any longer than the specified period. Otherwise, the contents of SRAM may be lost.
> If steps <1> to <3> may not be completed within 30 minutes, save all contents of the SRAM to the memory card or USB memory beforehand. Thus, if the contents of the SRAM are lost, the contents can be restored easily.
> For the methods of saving all contents and restoring them, refer to the maintenance manual (B-64485EN).

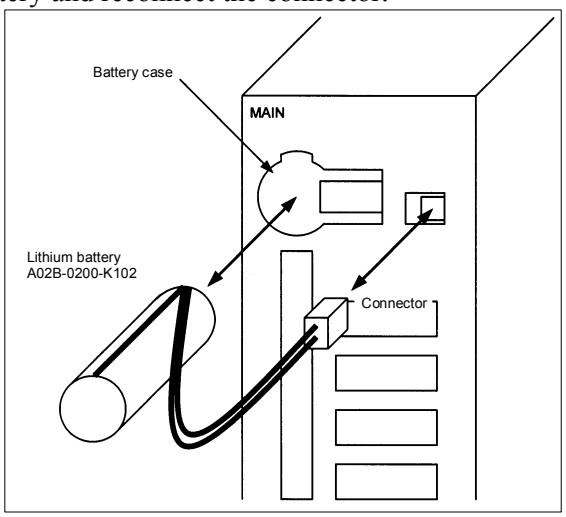
When using a lithium battery (for stand-alone type control unit)

Prepare a new lithium battery (ordering code: A02B-0200-K102).

<1> Turn the power to the machine (control unit) on. After about 30 seconds, turn the power off.

<2> Remove the lithium battery on the upper section of the control unit.
First, unplug the connector by yanking the battery cable, then take the battery out of its case.
The battery case is located in the upper section of the face plate of the main board.

<3> Insert a new lithium battery and reconnect the connector.



> ⚠**WARNING**
> Using other than the recommended battery may result in the battery exploding.
> Replace the battery only with the specified battery (A02B-0200-K102).

> ⚠ **CAUTION**
> Steps <1> to <3> should be completed within 30 minutes.
> Do not leave the control unit without a battery for any longer than the specified period. Otherwise, the contents of SRAM may be lost.
> If steps <1> to <3> may not be completed within 30 minutes, save all contents of the SRAM to the memory card or USB memory beforehand. Thus, if the contents of the SRAM are lost, the contents can be restored easily.
> For the methods of saving all contents and restoring them, refer to the maintenance manual (B-64485EN).

When using commercial D-size alkaline dry cells

<1> Have commercial D-size alkaline dry cells handy.

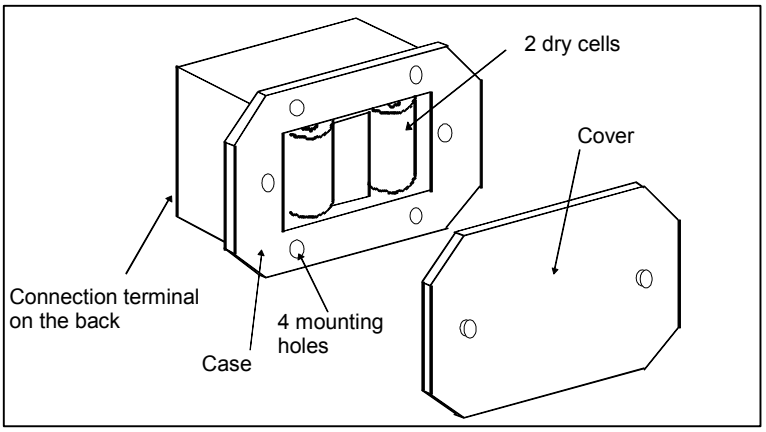
<2> Turn the power to the machine (control unit) on for about 30 seconds.

<3> Remove the cover from the battery case.

<4> Replace the old dry cells with new ones. Mount the dry cells in a correct orientation.

<5> Reinstall the cover onto the battery case.

> ⚠ **CAUTION**
> In the power-off state, the battery should be replaced as in the case of the lithium battery, which is descried above.



1.3.2 Battery in the PANEL *i* (3 VDC)

A lithium battery is used to back up BIOS data in the PANEL *i*. This battery is factory-set in the PANEL *i*. This battery has sufficient capacity to retain BIOS data for one year.
When the battery voltage becomes low, the following message appears on the self test screen displayed at turn-on, and the self test pauses.

CMOS Battery Failure

If this message appears, replace the battery as soon as possible (within one week). FANUC recommends that the battery be replaced once per year regardless of whether a battery alarm is issued.

- Replacing the battery

(1) To guard against the possible loss or destruction of BIOS parameters, write down the BIOS parameter values.
(2) Obtain a new lithium battery (A02B-0200-K102).
(3) After power has been supplied for at least five seconds, turn off the power to PANEL *i*. Remove the intelligent terminal from the panel so that replacement work can be done from the rear of the intelligent terminal.
(4) Detach the connector of the lithium battery, and remove the battery from the battery holder.
(5) Attach the connector, and place the battery in the battery holder.
(6) Install PANEL *i* again.
(7) Turn on the power, and check that the BIOS parameters are maintained (BIOS setup is not activated forcibly).

Between removing an old battery and inserting new battery, no more than five minutes must be allowed to elapse.

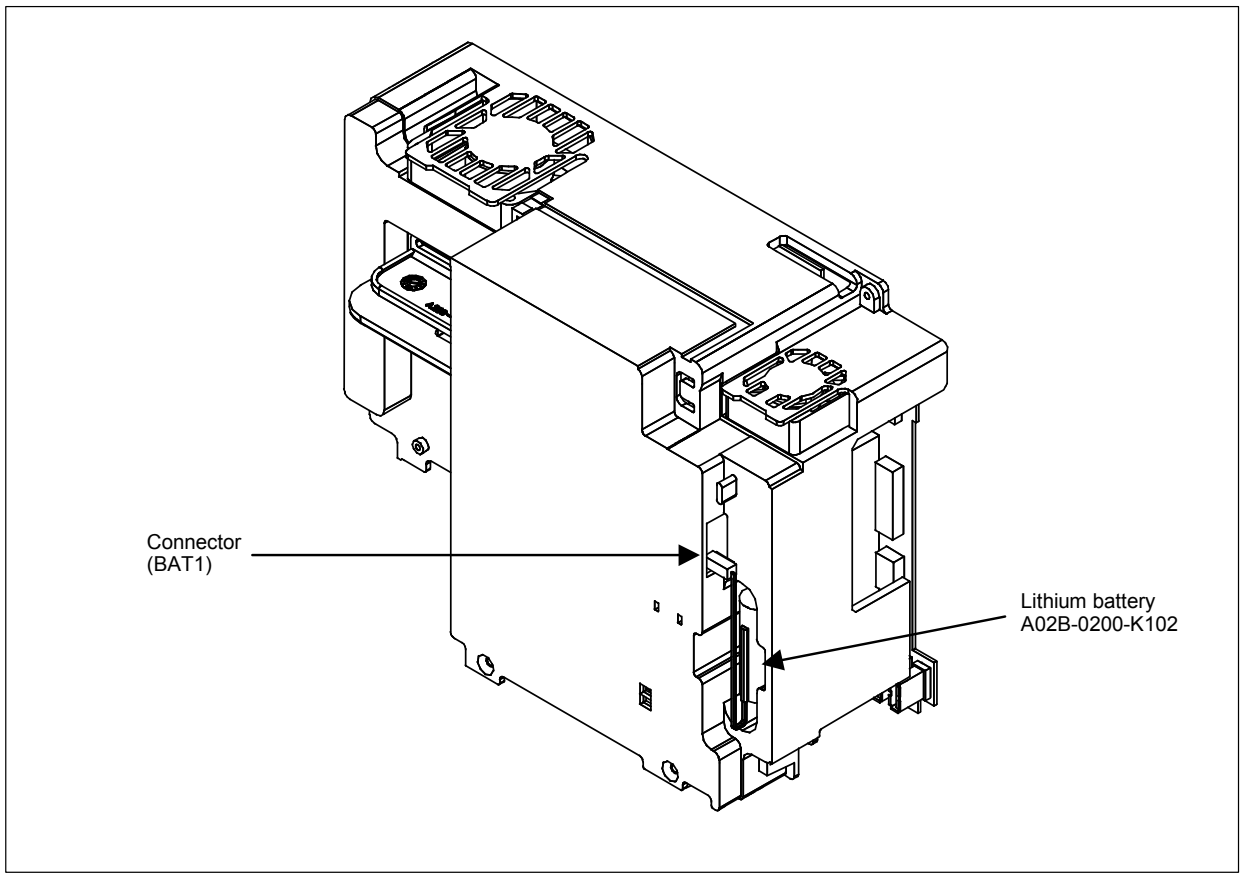


Fig. 1.3.2 (a) Lithium battery connection for PANEL *i*

1.3.3 Replacing Battery for Absolute Pulsecoders

1.3.3.1 Overview

- When the voltage of the batteries for absolute Pulsecoders becomes low, alarm 307 or 306 occurs, with the following indication in the CNC state display at the bottom of the CNC screen.
  Alarm 307 (alarm indicating the voltage of the battery becomes low) :
  The indication "APC" blinks in reversed display.
  Alarm 306 (battery zero alarm) :
  The indication "ALM" blinks in reversed display.
- When alarm 307 (alarm indicating the voltage of the battery becomes low) occurs, replace the battery as soon as possible. In general, the battery should be replaced within one or two weeks, however, this depends on the number of Pulsecoders used.
- When alarm 306 (battery zero alarm) occurs, Pulsecoders are reset to the initial state, in which absolute positions are not held. Alarm 300 (reference position return request alarm) also occurs, indicating that reference position return is required.
- In general, replace the batteries periodically within the service life listed below.
  - A06B-6050-K061 or D-size alkaline dry cells (LR20): Two years (for each six-axis configuration)
  - A06B-6073-K001 : Two years (for each three-axis configuration)
  - A06B-6114-K504 : One year (for each three-axis configuration)

NOTE
The above values indicate the estimated service life of batteries used with FANUC absolute Pulsecoders. The actual battery service life depends on the machine configuration based on, for example, detector types. For details, contact the machine tool builder.

1.3.3.2 Replacing batteries

To prevent absolute position information in absolute Pulsecoders from being lost, turn on the machine power before replacing the battery. The replacement procedure is described below.
<1> Ensure that the power to the servo amplifier is turned on.
<2> Ensure that the machine is in the emergency stop state (the motor is inactive).
<3> Ensure that the DC link charge LED of the servo amplifier is off.
<4> Detach the old batteries and attach new ones.
The replacement of the batteries in a separate battery case and the replacement of the battery built into the servo amplifier are described below in detail.

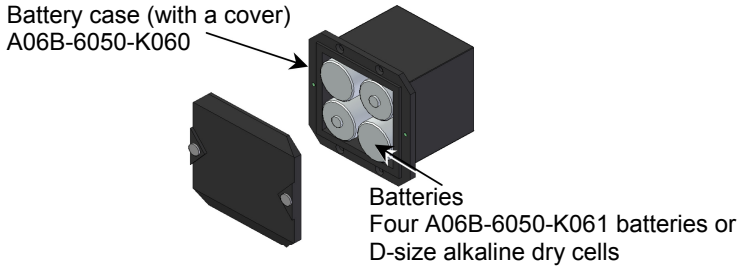
⚠ WARNING

- The absolute Pulsecoder of each of the $\alpha i/\alpha i$ S series servo motors and the βi S series servo motors (βi S0.4 to βi S22) has a built-in backup capacitor. Therefore, even when the power to the servo amplifier is off and the batteries are replaced, reference position return is not required if the replacement completes within less than 10 minutes. Turn the power on and replace the batteries if the replacement will take 10 minutes or more.
- To prevent electric shock, be careful not to touch metal parts in the power magnetics cabinet when replacing the batteries.
- Because the servo amplifier uses a large-capacitance electrolytic capacitor internally, the servo amplifier remains charged for a while even after the power is turned off. Before touching the servo amplifier for maintenance or other purposes, ensure your safety by measuring the residual voltage in the DC link with a tester and confirming that the charge indication LED (red) is off.
- Be sure to replace the batteries with specified ones. Pay attention to the battery polarity. If a wrong type of battery is used or a battery is installed with incorrect polarity, the battery may overheat, blow out, or catch fire, or the absolute position information in the absolute Pulsecoders may be lost.
- Ensure that the battery connector is inserted in the correct position.

1.3.3.3 Replacing the batteries in a separate battery case

Use the following procedure to replace the batteries in the battery case.

<1> Loosen the screws on the battery case and detach the cover.
<2> Replace the batteries in the case (pay attention to the polarity).
<3> Attach the cover to the battery case.



⚠ CAUTION

- Four D-size alkaline dry cells (LR20) that are commercially available can be used as batteries. A set of four A06B-6050-K061 batteries is optionally available from FANUC.
- Replace all the four batteries with new ones. If old and new batteries are mixed, the absolute position information in the absolute Pulsecoders may be lost.

1.3.3.4 Replacing the battery built into the servo amplifier

Use the following procedure to replace the special lithium battery.

<1> Detach the battery cover.
<2> Replace the special lithium battery.
<3> Attach the battery cover.

⚠ CAUTION

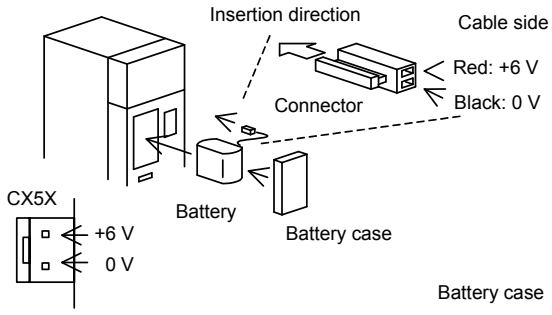
- Purchase the battery from FANUC because it is not commercially available. It is therefore recommended that you have a backup battery.
- When the built-in battery is used, do not connect BATL (B3) of connector CXA2A/CXA2B. Also, do not connect two or more batteries to the same BATL (B3) line. These connections are dangerous because battery output voltages may be short-circuited, causing the batteries to overheat.
- Install the battery in the servo amplifier in a direction that allows slack in the cable. If the battery cable is under tension, a bad connection may occur.
- If the +6 V pin and 0 V pin are short-circuited, the battery may overheat, blow out, or catch fire, or the absolute position information in the absolute Pulsecoders may be lost.
- When inserting the connector, align it to the connector pins.

[Connecting the battery]

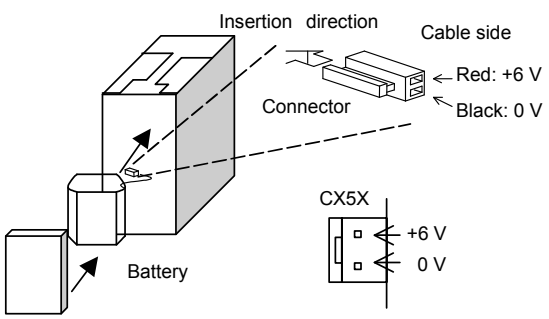
The battery for the β*i*SV4 and β*i*SV20 series amplifiers is mounted in the battery case on the underside of each of the amplifiers.

The battery for the other β*i* series amplifiers and the α*i* series amplifiers is mounted at the front of each of the amplifiers.

[α*i* series][β*i* series β*i* SV40, β*i* SV80]



[β*i* series β*i* SV4, β*i* SV 20]



[Battery sets and outlines]

Battery ordering drawing number	Manufacturer model number	Applicable servo amplifier	Battery case ordering drawing number	Outline
A06B-6114-K504 (Note)	BR-2/3AGCT4A (Panasonic)	α*i* series 60/90 mm width	A06B-6114-K505	
		α*i* series 150/300 mm width	A06B-6114-K506	
		β*i* series β*i* SV (two-axis model)	A06B-6114-K505	
A06B-6093-K001	BR-AGCF2W (Panasonic)	β*i* series β*i*SV4, β*i*SV20	A06B-6093-K002	
		β*i* series β*i*SV40, β*i*SV80	A06B-6093-K002	

NOTE

When an old type BR-CCF2TH battery is used, order a battery case that accommodates battery A06B-6114-K504.

Used batteries

Old batteries should be disposed as "INDUSTRIAL WASTES" according to the regulations of the country or autonomy where your machine has been installed.

APPENDIX

A PARAMETERS

This manual describes all parameters indicated in this manual.
For those parameters that are not indicated in this manual and other parameters, refer to the parameter manual.

NOTE
A parameter that is valid with only one of the path control types for the lathe system (T series) and machining center system (M series) is indicated in the upper or lower row as described below. A blank represents an unusable parameter.

[Example 1]
The parameter HTG is common to the T series and M series, and RTV and ROC are parameters used with the T series only.

	#7	#6	#5	#4	#3	#2	#1	#0	
1403	RTV		HTG	ROC					T series
			HTG						M series

[Example 2]
The following parameter is used with the M series only:

1411		T series
	Cutting feedrate	M series

A.1 DESCRIPTION OF PARAMETERS

	#7	#6	#5	#4	#3	#2	#1	#0
0000							ISO	TVC

[Input type] Setting input
[Data type] Bit path

#0 TVC TV check
0: Not performed
1: Performed

#1 ISO Code used for data output
0: EIA code
1: ISO code

NOTE
1 The I/O setting of a memory card is made by bit 0 (ISO) of parameter No. 0139.
2 The I/O setting of an USB memory is made by bit 0 (ISU) of parameter No. 11505.
3 The I/O setting of a data server is made by bit 0 (ISO) of parameter No. 0908.

NOTE
4 When EIA code is used for data output (ISO = 0), set bit 3 (ASI) of parameter No.101 and 111 and 121 to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
0001							FCV	

[Input type] Setting input
[Data type] Bit path

#1 FCV Program format
0: Series 16 standard format
1: Series 15 format

NOTE
1 Programs created in the Series 15 program format can be used for operation on the following functions:
1 Subprogram call M98
2 Thread cutting with equal leads G32 (T series)
3 Canned cycle G90, G92, G94 (T series)
4 Multiple repetitive canned cycle
G71 to G76 (T series) / G71.7 to G71.6 (M series)
5 Drilling canned cycle
G83.1, G80 to G89 (T series) / G73, G74, G76, G80 to G89 (M series)
2 When the program format used in the Series 15 is used for this CNC, some limits may add. Refer to the Operator's Manual.

	#7	#6	#5	#4	#3	#2	#1	#0
0010							PRM	

[Input type] Setting input
[Data type] Bit path

#1 PRM When parameters are output, the parameters whose values are 0 are:
0: Output.
1: Not output.

	#7	#6	#5	#4	#3	#2	#1	#0
0012								MIRx

[Input type] Setting input
[Data type] Bit axis

#0 MIRx Mirror image for each axis
0: Mirror image is off. (Normal)
1: Mirror image is on. (Mirror)

0020	I/O CHANNEL : Input/output device selection, or interface number for a foreground input device

[Input type] Setting input
[Data type] Byte
[Valid data range] 0 to 17

The CNC has the following interfaces for transferring data to and from an external input/output device and the host computer:

Input/output device interface (RS-232-C serial ports 1 and 2)
Memory card interface
Data server interface
Embedded Ethernet interface
USB memory interface

By setting bit 0 (IO4) of parameter No. 0110, data input/output can be controlled separately. When IO4 is not set, data input/output is performed using the channel set in parameter No. 0020. When IO4 is set, a channel can be assigned to each of foreground input, foreground output, background input, and background output.
In these parameters, specify the interface connected to each input/output device to and from which data is to be transferred. See the table below for these settings.

Correspondence between settings and input/output devices	
Setting	**Description**
0,1	RS-232-C serial port 1
2	RS-232-C serial port 2
4	Memory card interface
5	Data server interface
9	Embedded Ethernet interface
17	USB memory interface

	#7	#6	#5	#4	#3	#2	#1	#0
0100					NCR		CTV	

[Input type] Setting input
[Data type] Bit

#1 CTV Character counting for TV check in the comment section of a program.
0: Performed
1: Not performed

#3 NCR Output of the end of block (EOB) in ISO code
0: LF, CR, CR are output.
1: Only LF is output.

	#7	#6	#5	#4	#3	#2	#1	#0
0138	MNC							

[Input type] Parameter input
[Data type] Bit

#7 MNC DNC operation from the memory card and external device subprogram call from the memory card are:
0: Not performed.
1: Performed.

	#7	#6	#5	#4	#3	#2	#1	#0
0139								ISO

[Input type] Setting input
[Data type] Bit

#0 ISO When a memory card is selected as an I/O device, data input/output is performed using
0: ASCII codes.
1: ISO codes.

NOTE
See Appendix J, "ISO/ASCII code conversion tool" for conversion between ISO and ASCII code.

⚠ WARNING
1 Unless data is input using ASCII codes, set this parameter to 1 to input or output data using ISO codes.
2 Data input/output with ASCII codes is dangerous because parity information is not included and a data error during the data input/output is not detected.
3 DNC operation from a memory card also must set the parameter to 1, and execute DNC operation by ISO code. ASCII code is dangerous because parity information is not included and a data error during the data input is not detected.

	#7	#6	#5	#4	#3	#2	#1	#0
0908								ISO

[Input type] Setting input
[Data type] Bit

#0 ISO When a data server is selected as an I/O device, data input/output is performed using
0: ASCII codes.
1: ISO codes.

NOTE
See Appendix J, "ISO/ASCII code conversion tool" for conversion between ISO and ASCII code.

⚠ WARNING
1 Unless data is input using ASCII codes, set this parameter to 1 to input or output data using ISO codes.
2 Data input/output with ASCII codes is dangerous because parity information is not included and a data error during the data input/output is not detected.
3 DNC operation from a data server also must set the parameter to 1, and execute DNC operation by ISO code. ASCII code is dangerous because parity information is not included and a data error during the data input is not detected.

0983	Path control type of each path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input

[Data type] Byte path

[Valid data range] 0 to 1

Set the path control type of each path.
The following two path control types are available:
T series (lathe system) : 0
M series (machining system): 1

	#7	#6	#5	#4	#3	#2	#1	#0
1001								INM

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 INM Least command increment on the linear axis
0: In mm (metric system machine)
1: In inches (inch system machine)

	#7	#6	#5	#4	#3	#2	#1	#0
1002	IDG			XIK	AZR			JAX

[Input type] Parameter input
[Data type] Bit path

#0 JAX Number of axes controlled simultaneously in jog feed, manual rapid traverse and manual reference position return
0: 1 axis
1: 3 axes

#3 AZR When no reference position is set, the G28 command causes:
0: Reference position return using deceleration dogs (as during manual reference position return) to be executed.
1: Alarm PS0304, "G28 IS COMMANDED WITHOUT ZERO RETURN" to be displayed.

NOTE
When reference position return without dogs is specified, (when bit 1 (DLZ) of parameter No. 1005 is set to 1) the G28 command specified before a reference position is set causes an alarm PS0304 to be issued, regardless of the setting of AZR.

#4 XIK When bit 1 (LRP) of parameter No. 1401 is set to 0, namely, when positioning is performed using non-linear type positioning, if an interlock is applied to the machine along one of axes in positioning,
0: The machine stops moving along the axis for which the interlock is applied and continues to move along the other axes.
1: The machine stops moving along all the axes.

#7 IDG When the reference position is set without dogs, automatic setting of the bit 0 (IDGx) of parameter No. 1012 to prevent the reference position from being set again is:
0: Not performed.
1: Performed.

NOTE
When this parameter is set to 0, bit 0 (IDGx) of parameter No. 1012 is invalid.

	#7	#6	#5	#4	#3	#2	#1	#0
1004	IPR							

[Input type] Parameter input
[Data type] Bit path

#7 IPR When a number with no decimal point is specified, the least input increment of each axis is:
0: Not 10 times greater than the least command increment
1: 10 times greater than the least command increment
When the increment system is IS-A, and bit 0 (DPI) of parameter No. 3401 is set to 1 (pocket calculator type decimal point programming), the least input increment cannot be 10 times greater than the least command increment.

	#7	#6	#5	#4	#3	#2	#1	#0
1005			EDMx	EDPx				ZRNx

[Input type] Parameter input
[Data type] Bit axis

#0 ZRNx If a move command other than G28 is specified by automatic operation when no reference position return is performed yet after the power is turned on:
0: The alarm PS0224, "PERFORM REFERENCE POSITION RETURN." is issued.
1: Operation is performed without issuing an alarm.

NOTE
1 The state in which a reference position has not been established refers to the following state:
- When an absolute position detector is not used and reference position return has not been performed even once after power-up
- When an absolute position detector is used and the association of the machine position with the position detected with the absolute position detector has not been completed (See the description of bit 4 (APZx) of parameter No. 1815.)

2 When the Cs axis coordinates are to be set up, set ZRN to 0.

#4 EDPx In cutting feed, an external deceleration signal in the + direction for each axis is:
0: Invalid
1: Valid

#5 EDMx In cutting feed, an external deceleration signal in the - direction for each axis is:
0: Invalid
1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
1006			ZMIx		DIAx		ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROTx Setting linear or rotary axis.
#1 ROSx Setting linear or rotary axis.

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotary axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by bits 0 (ROAx) and 2 (RRLx) of parameter No. 1008. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No. 3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	1	Rotary axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624) (4) Cannot be used with the rotary axis roll-over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

#3 DIAx The move command for each axis is based on:
0: Radius specification
1: Diameter specification

#5 ZMIx The direction of manual reference position return is:
0: + direction
1: - direction

	#7	#6	#5	#4	#3	#2	#1	#0
1007			G90x		RAAx			

[Input type] Parameter input
[Data type] Bit axis

#3 RAAx Rotary axis control is:
0: Not performed.
1: Performed.
When an absolute programming is specified, the rotary axis control function determines the direction of rotation from the sign of the command value and determines an end coordinate from the absolute value of the command value.

NOTE
RAA is valid when bit 0 (ROA) of parameter No. 1008 is set to 1 and bit 1 (RAB) of parameter No. 1008 is set to 0.
To use this function, the option for rotary axis control is required.

#5 G90x A command for a rotary axis control is:

0: Regarded as an absolute/incremental programming according to the G90/G91 mode setting.

1: Regarded as an absolute programming at all times.

	#7	#6	#5	#4	#3	#2	#1	#0
1008						RRLx	RABx	ROAx

[Input type] Parameter input
[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ROAx The rotary axis roll-over is

0: Invalid

1: Valid

NOTE
ROAx specifies the function only for a rotary axis (for which bit 0 (ROTx) of parameter No. 1006 is set to 1)

#1 RABx In the absolute programming, the axis rotates in the direction

0: In which the distance to the target is shorter.

1: Specified by the sign of command value.

NOTE
RABx is valid only when ROAx is 1.

#2 RRLx Relative coordinates are

0: Not rounded by the amount of the shift per one rotation

1: Rounded by the amount of the shift per one rotation

NOTE
1 RRLx is valid only when ROAx is 1.
2 Assign the amount of the shift per one rotation in parameter No. 1260.

	#7	#6	#5	#4	#3	#2	#1	#0
1013					ISEx	ISDx	ISCx	ISAx

[Input type] Parameter input
[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ISAx
#1 ISCx
#2 ISDx
#3 ISEx Increment system of each axis

Increment system	Bit 3 (ISE)	Bit 2 (ISD)	Bit 1 (ISC)	Bit 0 (ISA)
IS-A	0	0	0	1
IS-B	0	0	0	0
IS-C	0	0	1	0
IS-D	0	1	0	0
IS-E	1	0	0	0

	#7	#6	#5	#4	#3	#2	#1	#0
1015	DWT							

[Input type] Parameter input
[Data type] Bit path

#7 DWT When time for dwell per second is specified by P, the increment system:
0: Depends on the increment system
1: Does not depend on the increment system (1 ms)

1020	Program axis name for each axis

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 65 to 67, 85 to 90

An axis name (axis name 1: parameter No. 1020) can be arbitrarily selected from A, B, C, U, V, W, X, Y, and Z. (When G code system A is used with the lathe system, however, U, V, and W are not selectable.) When bit 0 (EEA) of parameter No. 1000 is set to 1, the length of an axis name can be extended to three characters by setting axis name 2 (parameter No. 1025) and axis name 3 (parameter No. 1026) (extended axis name).
For axis names 2 and 3, a character from 0 to 9 and A to Z of ASCII code can be arbitrarily selected. However, the setting of axis name 3 for each axis is invalid if axis name 2 is not set. Moreover, if a character from 0 to 9 is set as axis name 2, do not use a character from A to Z as axis name 3.

(Tip) ASCII code

Axis name	X	Y	Z	A	B	C	U	V	W
Setting	88	89	90	65	66	67	85	86	87

When G code system A is used with the lathe system, and the character X, Y, Z, or C is used as axis name 1 of an axis, a command with U, V, W, or H specified for axis name 1 represents an incremental programming for the axis.

NOTE
1 When a multiple repetitive canned cycle for turning is used, no character other than X, Y, and Z can be used as the address of the axis.
2 An address other than addresses A, B, and C cannot be used as the address of a rotary axis used for the function for tool length compensation in a specified direction or the tool center point control function.

NOTE
3 When the custom macro function is enabled, the same extended axis name as a reserved word cannot be used. Such an extended axis name is regarded as a reserved word.
Because of reserved words of custom macros, extended axis names that start with the following two characters cannot be used: AB, AC, AD, AN, AS, AT, AX, BC, BI, BP, CA, CL, CO, US, WH, WR, XO, ZD, ZE, ZO, ZW
4 In a macro call, no extended axis name can be used as an argument.

1022 **Setting of each axis in the basic coordinate system**

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7

To determine a plane for circular interpolation, cutter compensation, and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane, G19: Yp-Zp plane) and a 3-dimensional tool compensation space (XpYpZp), specify which of the basic three axes (X, Y, and Z) is used for each control axis, or a parallel axis of which basic axis is used for each control axis.
A basic axis (X, Y, or Z) can be specified only for one control axis.
Two or more control axes can be set as parallel axes for the same basic axis.

Setting	Meaning
0	Rotary axis (Neither the basic three axes nor a parallel axis)
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

In general, the increment system and diameter/radius specification of an axis set as a parallel axis are to be set in the same way as for the basic three axes.

1023 **Number of the servo axis for each axis**

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 80

This parameter associates each control axis with a specific servo axis. Specify values 1+8n, 2+8n, 3+8n, 4+8n, 5+8n, and 6+8n (n = 0, 1, 2, …, 9) like 1, 2, 3, 4, 5, …, 77, and 78.
The control axis number is the order number that is used for setting the axis-type parameters or axis-type machine signals

- With an axis for which Cs contour control/spindle positioning is to be performed, set -(spindle number) as the servo axis number.

Example)
When exercising Cs contour control on the fourth controlled axis by using the first spindle, set -1.

- For tandem controlled axes or electronic gear box (EGB) controlled axes, two axes need to be specified as one pair. So, make a setting as described below.
  Tandem axis: For a master axis, set an odd (1, 3, 5, 7, 9, ...) servo axis number. For a slave axis to be paired, set a value obtained by adding 1 to the value set for the master axis.
  EGB axis: For a slave axis, set an odd (1, 3, 5, 7, 9, ...) servo axis number. For a dummy axis to be paired, set a value obtained by adding 1 to the value set for the slave axis.

1025	Program axis name 2 for each axis
1026	Program axis name 3 for each axis

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 48 to 57, 65 to 90
When axis name extension is enabled (when bit 0 (EEA) of parameter No. 1000 is set to 1), the length of an axis name can be extended to a maximum of three characters by setting axis name 2 and axis name 3.

NOTE
If program axis name 2 is not set, program axis name 3 is invalid.

1031	Reference axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes
The unit of some parameters common to all axes such as those for dry run feedrate and single-digit F1 feedrate may vary according to the increment system. An increment system can be selected by a parameter on an axis-by-axis basis. So, the unit of those parameters is to match the increment system of a reference axis. Set which axis to use as a reference axis.
Among the basic three axes, the axis with the finest increment system is generally selected as a reference axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1201					FPC	ZCL		ZPR

[Input type] Parameter input
[Data type] Bit path

#0 ZPR Automatic setting of a coordinate system when the manual reference position return is performed
0: Not set automatically
1: Set automatically

NOTE
ZPR is valid while a workpiece coordinate system function is not provided. If a workpiece coordinate system function is provided, making a manual reference position return always causes the workpiece coordinate system to be established on the basis of the workpiece zero point offset (parameters Nos. 1220 to 1226), irrespective of this parameter setting.

#2 ZCL Local coordinate system when the manual reference position return is performed
0: The local coordinate system is not canceled.
1: The local coordinate system is canceled.

NOTE
ZCL is valid when the workpiece coordinate system option is specified. In order to use the local coordinate system (G52), the workpiece coordinate system option is required.

#3 FPC When a floating reference position is set with a soft key, the relative position indication is:
0: Not preset to 0 (The relative position indication remains unchanged.)
1: Preset to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
1202						G92		

[Input type] Parameter input
[Data type] Bit path

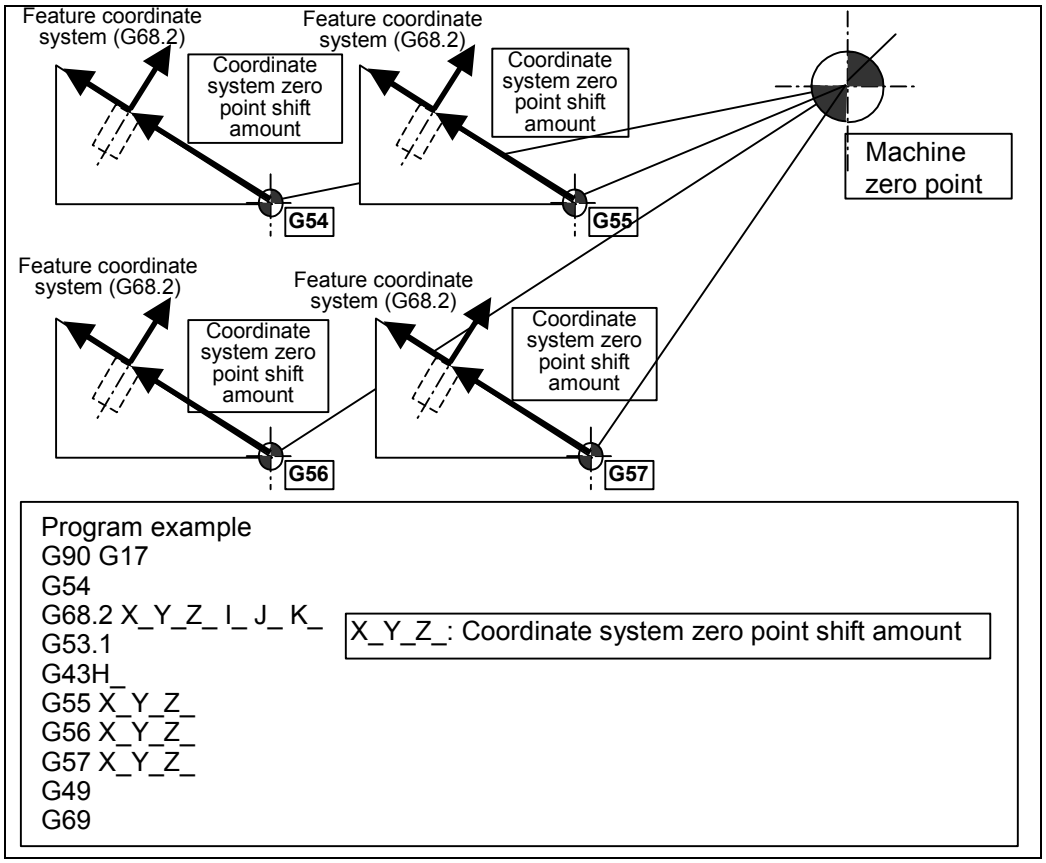
#2 G92 When the CNC has commands G52 to G59 specifying workpiece coordinate systems (optional function), if the G command for setting a coordinate system (G92 for M series, G50 for T series (or the G92 command in G command system B or C)) is specified,
0: G command is executed and no alarm is issued.
1: G command is not executed and an alarm PS0010, “IMPROPER G-CODE” is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
1205		3TW						

[Input type] Parameter input
[Data type] Bit path

#6 3TW When workpiece coordinate system selection is specified with G code in tilted working plane command mode:
0: The alarm PS5462, "ILLEGAL COMMAND (G68.2/G69)" is issued.
1: Workpiece coordinate system selection is executed.

⚠ CAUTION
When this parameter is 1, only G54 to G59 or G54.1 can be specified. Specifying G52 or G92 causes alarm PS5462. Specifying G54 to G59 or G54.1 suppresses buffering.



1240	Coordinate value of the reference position in the machine coordinate system

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the reference position in the machine coordinate system.

1241	Coordinate value of the second reference position in the machine coordinate system
1242	Coordinate value of the third reference position in the machine coordinate system
1243	Coordinate value of the fourth reference position in the machine coordinate system

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the second to fourth reference positions in the machine coordinate system.

1244	Coordinate value of the floating reference position in the machine coordinate system

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate values of the floating reference position in the machine coordinate system.

1250	Coordinate system of the reference position used when automatic coordinate system setting is performed

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate system of the reference position on each axis to be used for performing automatic coordinate system setting.

1260	The shift amount per one rotation of a rotary axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
Set the shift amount per one rotation of a rotary axis.
For the rotary axis used for cylindrical interpolation, set the standard value.

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA					LMS	NAL	OUT

[Input type] Setting input
[Data type] Bit path

#0 OUT The area inside or outside of the stored stroke check 2 is set as an inhibition area
0: Inside
1: Outside

#1 NAL When the tool enters the inhibition area of stored stroke limit 1:
0: The overtravel alarm signal is not output.
1: The overtravel alarm signal is output, and the tool is decelerated to a stop.
If manual operation is in progress at this time, the alarm is not output.

NOTE
When this parameter is set to 1, the alarm is issued if the tool enters stored stroke limit 1 during automatic operation.

#2 LMS The stored stroke check 1 select signal (EXLM3, EXLM2, or EXLM when stored stroke check 1 area expansion is used) for switching stored stroke check
0: Disabled
1: Enabled

NOTE
When bit 0 (DLM) of parameter No. 1301 is set to 1, the stored stroke check 1 select signal EXLM <Gn007.6> (EXLM3 <Gn531.7>, EXLM2 <Gn531.6>, or EXLM <Gn007.6> when stored stroke check 1 area expansion is used) is made invalid.

#7 BFA When the stored stroke check 1, 2, or 3 alarm is issued, an interference alarm is issued with the inter-path interference check function (T series), or a chuck/tail stock barrier (T series) alarm is issued:
0: The tool stops after entering the prohibited area.
1: The tool stops before the prohibited area.

	#7	#6	#5	#4	#3	#2	#1	#0
1301		OTS				NPC		

[Input type] Setting input
[Data type] Bit path

#2 NPC As part of the stroke limit check performed before movement, the movement specified in G31 (skip) and G37 (automatic tool length measurement) blocks is:
0: Checked
1: Not checked

NOTE
This parameter is valid only when the option for stroke limit check before movement is selected.

#6 OTS When the overtravel alarm is issued:
0: The overtravel alarm signal is not output to the PMC.
1: The overtravel alarm signal is output to the PMC.

1320	Coordinate value I of stored stroke check 1 in the positive direction on each axis
1321	Coordinate value I of stored stroke check 1 in the negative direction on each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate value of stored stroke check 1 on each axis in the + or - direction in the machine coordinate system.

NOTE
1 Specify diameter values for any axes for which diameter programming is specified.
2 The area outside the area set by parameters Nos. 1320 and 1321 is a prohibited area.

1322	Coordinate value of stored stroke check 2 in the positive direction on each axis
1323	Coordinate value of stored stroke check 2 in the negative direction on each axis

[Input type] Setting input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate value of stored stroke check 2 on each axis in the + or - direction in the machine coordinate system.

NOTE
1 Specify diameter values for any axes for which diameter programming is specified.
2 Whether the inside area or outside area is a prohibited area is set using bit 0 (OUT) of parameter No. 1300.

1324	Coordinate value of stored stroke check 3 in the positive direction on each axis
1325	Coordinate value of stored stroke check 3 in the negative direction on each axis

[Input type] Setting input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate value of stored stroke check 3 on each axis in the + or - direction in the machine coordinate system.

NOTE
1 Specify diameter values for any axes for which diameter programming is specified.
2 The area inside the area set by parameters Nos. 1324 and 1325 is a prohibited area.

1326	Coordinate value II of stored stroke check 1 in the positive direction on each axis
1327	Coordinate value II of stored stroke check 1 in the negative direction on each axis
1350	Coordinate value III of stored stroke check 1 in the positive direction on each axis
1351	Coordinate value III of stored stroke check 1 in the negative direction on each axis
1352	Coordinate value IV of stored stroke check 1 in the positive direction on each axis

1353	Coordinate value IV of stored stroke check 1 in the negative direction on each axis
1354	Coordinate value V of stored stroke check 1 in the positive direction on each axis
1355	Coordinate value V of stored stroke check 1 in the negative direction on each axis
1356	Coordinate value VI of stored stroke check 1 in the positive direction on each axis
1357	Coordinate value VI of stored stroke check 1 in the negative direction on each axis
1358	Coordinate value VII of stored stroke check 1 in the positive direction on each axis
1359	Coordinate value VII of stored stroke check 1 in the negative direction on each axis
1360	Coordinate value VIII of stored stroke check 1 in the positive direction on each axis
1361	Coordinate value VIII of stored stroke check 1 in the negative direction on each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate value of stored stroke check 1 on each axis in the + or - direction in the machine coordinate system.
The stored stroke check 1 select signal (EXLM3, EXLM2, EXLM) and the stroke parameter to be selected have the following relationships:

EXLM3	EXLM2	EXLM	Stroke parameter to be selected
0	0	0	Coordinate value I (No. 1320 / No. 1321)
0	0	1	Coordinate value II (No. 1326 / No. 1327)
0	1	0	Coordinate value III (No. 1350 / No. 1351)
0	1	1	Coordinate value IV (No. 1352 / No. 1353)
1	0	0	Coordinate value V (No. 1354 / No. 1355)
1	0	1	Coordinate value VI (No. 1356 / No. 1357)
1	1	0	Coordinate value VII (No. 1358 / No. 1359)
1	1	1	Coordinate value VIII (No. 1360 / No. 1361)

NOTE
1 Specify diameter values for any axes for which diameter programming is specified.
2 The outside of the area set with each parameter is treated as the inhibition area.
3 The stored stroke check 1 select signal (EXLM3, EXLM2, EXLM) is valid only when bit 2 (LMS) of parameter No. 1300 is 1.
4 When axis direction dependent stored stroke check 1 is enabled (with bit 0 (DLM) of parameter No. 1301 set to 1), stroke parameter switching by the stored stroke check 1 select signal (EXLM3, EXLM2, EXLM) is disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
1401				RF0			LRP	RPD

[Input type] Parameter input
[Data type] Bit path

#0 RPD Manual rapid traverse during the period from power-on time to the completion of the reference position return.
0: Disabled (Jog feed is performed.)
1: Enabled

#1 LRP Positioning (G00)
0: Positioning is performed with non-linear type positioning so that the tool moves along each axis independently at rapid traverse.
1: Positioning is performed with linear interpolation so that the tool moves in a straight line.
When using 3-dimensional coordinate system conversion, set this parameter to 1.

#4 RF0 When cutting feedrate override is 0% during rapid traverse,
0: The machine tool does not stop moving.
1: The machine tool stops moving.

	#7	#6	#5	#4	#3	#2	#1	#0
1402				JRV				NPC

[Input type] Parameter input
[Data type] Bit path

#0 NPC Feed per revolution without the position coder (function for converting feed per revolution F to feed per minute F in the feed per revolution mode (G95)) is:
0: Not used
1: Used

#4 JRV Jog feed or incremental feed is
0: Performed at feed per minute.
1: Performed at feed per rotation.

NOTE
1 Specify a feedrate in parameter No. 1423.
2 For the machining center system, the option for threading/synchronous feed is required.

	#7	#6	#5	#4	#3	#2	#1	#0
1403			HTG					

[Input type] Parameter input
[Data type] Bit path

#5 HTG The feedrate for helical interpolation/helical involute interpolation/3-dimensional circular interpolation is:
0: Specified using the feedrate along the tangent to an arc/involute curve/3-dimensional arc
1: Specified using the feedrate along axes including a linear axis (specified axes other than the circular interpolation axis in the case of 3-dimensional circular interpolation)

	#7	#6	#5	#4	#3	#2	#1	#0
1404	FC0					FM3		
	FC0							

[Input type] Parameter input
[Data type] Bit path

#2 FM3 The increment system of an F command without a decimal point in feed per minute is:
0: 1 mm/min (0.01 inch/min for inch input)
1: 0.001 mm/min (0.00001 inch/min for inch input)

#7 FC0 Specifies the behavior of the machine tool when a block (G01, G02, G03, etc.) containing a feedrate command (F command) that is 0 is issued during automatic operation, as follows:
0: An alarm PS0011, “FEED ZERO (COMMAND)” occurs.
1: An alarm PS0011 does not occur, and the block is executed.

NOTE
1 In inverse time feed (G93) mode, the alarm PS1202, "NO F COMMAND AT G93" is issued irrespective of the setting of this parameter.
2 This parameter is set from 1 to 0, if a bit 6 (CLR) of parameter No. 3402 is 1, reset the CNC. Or if CLR is 0, turn off and on the CNC.

	#7	#6	#5	#4	#3	#2	#1	#0
1405								
							FR3	

[Input type] Parameter input
[Data type] Bit path

#1 FR3 The increment system of an F command without a decimal point in feed per revolution is:
0: 0.01 mm/rev (0.0001 inch/rev for inch input)
1: 0.001 mm/rev (0.00001 inch/rev for inch input)

	#7	#6	#5	#4	#3	#2	#1	#0
1408								RFDx

[Input type] Parameter input
[Data type] Bit axis

#0 RFDx Feedrate control on a rotary axis is exercised using:
0: Conventional method
1: Method that specifies a feedrate on the virtual circle of the rotary axis

1410	Dry run rate

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set the dry run rate at the 100% position on the jog feedrate specification dial. The unit of data depends on the increment system of the reference axis.

1411	
	Cutting feedrate

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
A cutting feedrate can be specified with this parameter for a machine which does not have to change the cutting feedrate frequently during machining. This eliminates the need to specify a cutting feedrate (F code) in the NC program.

1414	
	Feedrate for retrace

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set a cutting feedrate for retrace operation. When 0 is set, a retrace operation is performed at a programmed feedrate.

NOTE
This parameter has no effect on blocks that are in rapid traverse mode during reverse execution.

1420	**Rapid traverse rate for each axis**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1421	**F0 rate of rapid traverse override for each axis**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set the F0 rate of the rapid traverse override for each axis.

1423	Feedrate in manual continuous feed (jog feed) for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

(1) When bit 4 (JRV) of parameter No. 1402 is set to 0 (feed per minute), specify a jog feedrate (feed per minute) under an override of 100%.
(2) When bit 4 (JRV) of parameter No. 1402 is set to 1 (feed per revolution), specify a jog feedrate (feed per revolution) under an override of 100%.

NOTE
This parameter is clamped to the axis-by-axis manual rapid traverse rate (parameter No. 1424).

1424	Manual rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the rate of manual rapid traverse when the rapid traverse override is 100% for each axis.

NOTE
1 If 0 is set, the rate set in parameter No. 1420 (rapid traverse rate for each axis) is assumed.
2 When manual rapid traverse is selected (bit 0 (RPD) of parameter No. 1401 is set to 1), manual feed is performed at the feedrate set in this parameter, regardless of the setting of bit 4 (JRV) of parameter No. 1402.

1425	FL rate of the reference position return for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set feedrate (FL rate) after deceleration when the reference position return is performed for each axis.

1427	External deceleration rate of rapid traverse for each axis

[Input type] Parameter input
[Data type] Real axis

[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the external deceleration rate of rapid traverse for each axis.

1428	Reference position return feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

This parameter sets a rapid traverse rate for reference position return operation using deceleration dogs, or for reference position return operation before a reference position is set.
This parameter is also used to set a feedrate for the rapid traverse command (G00) in automatic operation before a reference position is set.

NOTE

1 To this feedrate setting 100%, a rapid traverse override (F0, 25, 50, or 100%) is applicable.

2 For automatic return after completion of reference position return and machine coordinate system establishment, the normal rapid traverse rate is used.

3 As a manual rapid traverse rate before machine coordinate system establishment by reference position return, the jog feedrate or manual rapid traverse rate can be selected with bit 0 (RPD) of parameter No. 1401.

	Before coordinate system establishment	After coordinate system establishment
Automatic reference position return (G28)	No. 1428	No. 1420
Automatic rapid traverse (G00)	No. 1428	No. 1420
Manual reference position return *1	No. 1428	No. 1428 *3
Manual rapid traverse	No. 1423 *2	No. 1424

4 When parameter No. 1428 is set to 0, the following parameter-set feedrates are applied.

	Before coordinate system establishment	After coordinate system establishment
Automatic reference position return (G28)	No. 1420	No. 1420
Automatic rapid traverse (G00)	No. 1420	No. 1420
Manual reference position return *1	No. 1424	No. 1424 *3
Manual rapid traverse	No. 1423 *2	No. 1424

No. 1420: Rapid traverse rate
No. 1423: Jog feedrate
No. 1424: Manual rapid traverse rate

*1 : By using bit 2 (JZR) of parameter No. 1401, the jog feedrate can be used for manual reference position return at all times.

*2 : When bit 0 (RPD) of parameter No. 1401 is set to 1, the setting of parameter No. 1424 is used.

*3 : When rapid traverse is used for reference position return without dogs or manual reference position return after reference position establishment, regardless of the deceleration dog, the feedrate for manual reference position return based on these functions is used (the setting of bit 1 (DLF) of parameter No. 1404 is followed).

1430 **Maximum cutting feedrate for each axis**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Specify the maximum cutting feedrate for each axis.

1432 **Maximum cutting feedrate for all axes in the acceleration/deceleration before interpolation**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)

Set a maximum cutting feedrate for each axis in the acceleration/deceleration before interpolation mode such as AI contour control. When the acceleration/deceleration before interpolation mode is not set, the maximum cutting feedrate set in parameter No. 1430 is used.

1434	Maximum manual handle feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set a maximum manual handle feedrate for each axis in case of maximum manual handle feedrate switch signal HNDLF<Gn023.3>=”1”.

1441	External deceleration rate setting 2 for each axis in rapid traverse

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set external deceleration rate 2 for each axis in rapid traverse.

1444	External deceleration rate setting 3 for each axis in rapid traverse

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set external deceleration rate 3 for each axis in rapid traverse.

1445	Maximum manual handle feedrate setting 3 for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set a maximum manual handle feedrate 3 for each axis.

1450	Change of feedrate for one graduation on the manual pulse generator during one-digit F feed code

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 127
Set the constant that determines the change in feedrate as the manual pulse generator is rotated one graduation during one-digit F feed code.

$$\Delta F = \frac{F \max i}{100n} \quad \text{(where, i=1 or 2)}$$

In the above equation, set n. That is, the number of revolutions of the manual pulse generator, required to reach feedrate Fmaxi is obtained. Fmaxi refers to the upper limit of the feedrate for a one-digit F code feed command, and set it in parameter No. 1460 or 1461.
Fmax1: Upper limit of the feedrate for F1 to F4 (parameter No. 1460)
Fmax2: Upper limit of the feedrate for F5 to F9 (parameter No. 1461)

1451	Feedrate for F1
to	to
1459	Feedrate for F9

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
These parameters set the feedrates for one-digit F code feed commands F1 to F9. When a one-digit F code feed command is specified, and the feedrate is changed by turning the manual pulse generator, the parameter-set value also changes accordingly.

1460	Upper limit of feedrate for F1 to F4
1461	Upper limit of feedrate for F5 to F9

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set the upper limit of feedrate for the one-digit F code feed command.
As the feedrate increases by turning the manual pulse generator, the feedrate is clamped when it reaches the upper limit set. If a one-digit F feed command F1 to F4 is executed, the upper limit is that set in parameter No. 1460. If a one-digit F code feed command F5 to F9 is executed, the upper limit is that set in parameter No. 1461.

1465	Radius of a virtual circle when a feedrate is specified on the virtual circle of a rotary axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (B)
Set the radius of a virtual circle when a feedrate on the virtual circle of a rotary axis is specified.
If 0 is set for a rotary axis, the axis is excluded from feedrate calculation.
If the input unit is the inch, enter a value in inches.
The data is then converted to a millimeter value and displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
1490	PGF							

[Input type] Parameter input
[Data type] Bit path

#7 PGF The feedrate specified for circular interpolation, involute interpolation, spiral/conical interpolation, and NURBS interpolation in the high-speed program check mode is:

0: The dry run feedrate.
At this time, manual feedrate override signals *JV0 to *JV15 <Gn010 to Gn011> can be used.

1: The maximum feedrate specified by the CNC.

> **⚠ CAUTION**
> If this parameter is set to 1, feedrate clamp, override, and dry run for circular interpolation, involute interpolation, spiral/conical interpolation, and NURBS interpolation are disabled. If a movement around a stroke limit is specified, therefore, a stroke limit check cannot sometimes be made correctly.

	#7	#6	#5	#4	#3	#2	#1	#0
1601			NCI					

[Input type] Parameter input
[Data type] Bit path

#5 NCI An in-position check:

0: Confirms that the specified feedrate becomes 0 (the acceleration/deceleration delay becomes 0) at deceleration time and that the machine position has reached a specified position (the servo positional deviation is within the in-position width set by parameter No. 1826).

1: Confirms only that the specified feedrate becomes 0 (the acceleration/deceleration delay becomes 0) at deceleration time.

	#7	#6	#5	#4	#3	#2	#1	#0
1602						CAF		

[Input type] Parameter input
[Data type] Bit path

#2 CAF In cutter compensation mode or tool nose radius compensation mode, the circular cutting point feedrate change is;

0: Performed on the inner arc only.

1: Performed on the inner and outer arcs.

	#7	#6	#5	#4	#3	#2	#1	#0
1604								SHP

[Input type] Parameter input
[Data type] Bit path

#0 SHP When automatic operation is started, the state equivalent to the specification of G5.1Q1 for AI contour control is:
0: Not set
1: Set
Upon reset, the state where G5.1Q1 is specified is set.

	#7	#6	#5	#4	#3	#2	#1	#0
1606								MNJx

[Input type] Parameter input
[Data type] Bit axis

#0 MNJx In manual handle interrupt or automatic manual simultaneous operation (interrupt type):
0: Only cutting feed acceleration/deceleration is enabled, and jog feed acceleration/deceleration is disabled.
1: Both cutting feed acceleration/deceleration and jog feed acceleration/deceleration are applied.

	#7	#6	#5	#4	#3	#2	#1	#0
1610							CTBx	CTLx

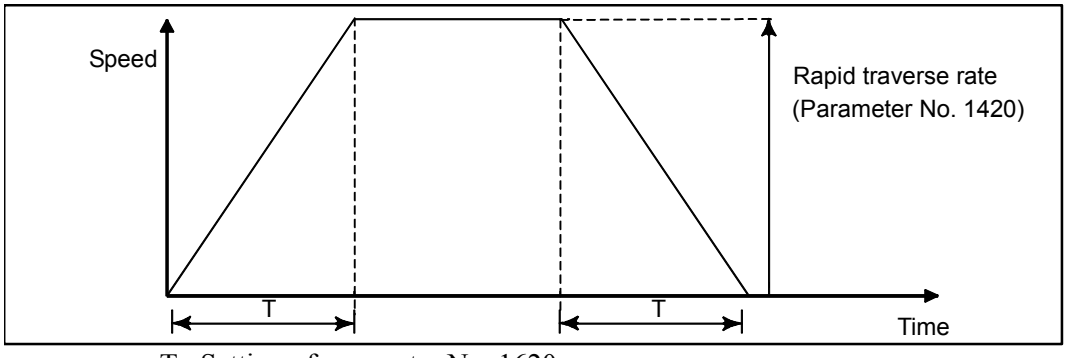
[Input type] Parameter input
[Data type] Bit axis

#0 CTLx Acceleration/deceleration in cutting feed or dry run during cutting feed
0: Exponential acceleration/deceleration is applied.
1: Linear acceleration/deceleration after interpolation is applied.

#1 CTBx Acceleration/deceleration in cutting feed or dry run during cutting feed
0: Exponential acceleration/deceleration or linear acceleration/ deceleration is applied. (depending on the setting in bit 0 (CTLx) of parameter No. 1610)
1: Bell-shaped acceleration/deceleration is applied.

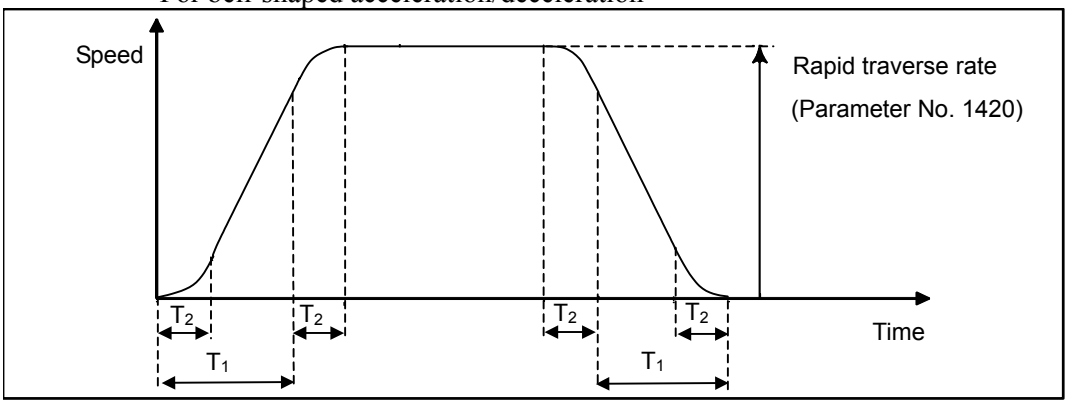
1620	Time constant T or T1 used for linear acceleration/deceleration or bell-shaped acceleration/deceleration in rapid traverse for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
Specify a time constant used for acceleration/deceleration in rapid traverse.
[Example]
For linear acceleration/deceleration



T : Setting of parameter No. 1620

For bell-shaped acceleration/deceleration



T_1 : Setting of parameter No. 1620
T_2 : Setting of parameter No. 1621
(However, $T_1 \geq T_2$ must be satisfied.)
Total acceleration (deceleration) time : $T_1 + T_2$
Time for linear portion : $T_1 - T_2$
Time for curve portion : $T_2 \times 2$

1622 **Time constant of acceleration/deceleration in cutting feed for each axis**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in cutting feed, bell-shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation in cutting feed for each axis. Which type to use is selected with bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610. Except for special applications, the same time constant must be set for all axes in this parameter. If the time constants set for the axes differ from each other, proper straight lines and arcs cannot be obtained.

1624 **Time constant of acceleration/deceleration in jog feed for each axis.**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set the time constant used for acceleration/deceleration in jog feed for each axis.

1660 **Maximum allowable acceleration rate in acceleration/deceleration before interpolation for each axis**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, $inch/sec^2$, $degree/sec^2$ (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)

(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)

Set a maximum allowable acceleration rate in acceleration/ deceleration before interpolation for each axis.

If a value greater than 100000.0 is set, the value is clamped to 100000.0.

If 0 is set, the specification of 100000.0 is assumed. If 0 is set for all axes, however, acceleration/deceleration before interpolation is not performed.

If a maximum allowable acceleration rate set for one axis is greater than a maximum allowable acceleration rate set for another axis by a factor or 2 or more, the feedrate at a corner where the direction of travel abruptly changes can decrease temporarily.

1671	**Maximum allowable acceleration rate in acceleration/deceleration before interpolation for linear rapid traverse for each axis, or maximum allowable reference acceleration rate in optimum torque acceleration/deceleration**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, degree/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)

(1) Set a maximum allowable acceleration rate in acceleration/ deceleration before interpolation for linear rapid traverse.
If a value greater than 100000.0, the value is clamped to 100000.0.
If 0 is set, the specification of the following is assumed:
1000.0 mm/sec^2
100.0 inch/sec^2
100.0 degrees/sec^2
If 0 is specified for all axes, however, acceleration/deceleration before interpolation is not performed.

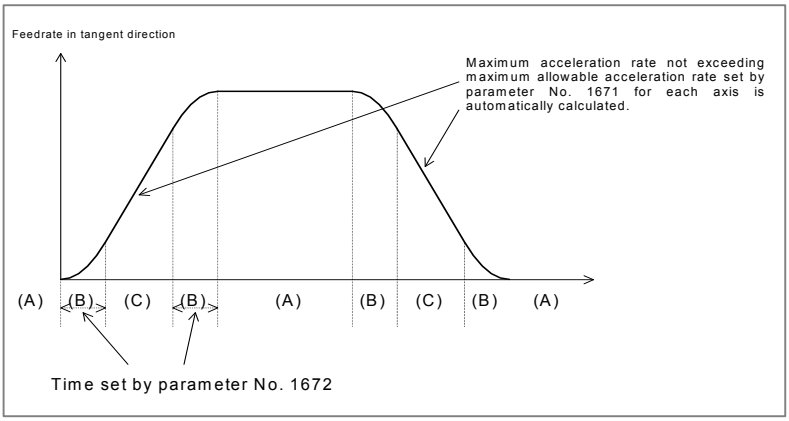
(2) Maximum allowable reference acceleration rate in optimum torque acceleration/deceleration

1672	**Acceleration change time of bell-shaped acceleration/deceleration before interpolation for linear rapid traverse, or acceleration change time of bell-shaped acceleration/deceleration in optimum torque acceleration/deceleration**

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 200

(1) Set an acceleration change time of bell-shaped acceleration/ deceleration for linear rapid traverse (time for changing from the state of constant feedrate (A) to the state of constant acceleration/deceleration (C) at the acceleration rate calculated from the acceleration rate set in parameter No. 1671: time of (B) in the figure below).

(2) Set an acceleration change time of bell-shaped acceleration/ deceleration in optimum torque acceleration/deceleration (time for changing from the state of constant feedrate (A) to the state of acceleration/deceleration (C) at the acceleration rate calculated from optimum torque acceleration/deceleration: time of (B) in the figure).



1710	Minimum deceleration ratio (MDR) for inner circular cutting feedrate change by automatic corner override

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 100

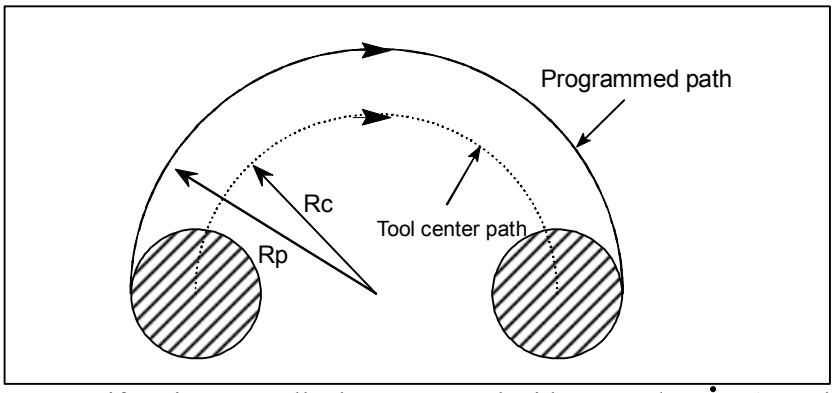
Set a minimum deceleration ratio (MDR) for an inner circular cutting feedrate change by automatic corner override.

In the case of circular cutting offset inward, the actual feedrate is determined by a specified feedrate (F) as follows:

$$F \times \frac{Rc}{Rp}$$

Rc: Radius of tool center path
Rp: Programmed radius

Thus, the feedrate along the programmed path satisfies the specified value of F.



However, if Rc is too small when compared with Rp, Rc/Rp $\fallingdotseq$ 0 results to stop the tool. So, a minimum deceleration ratio (MDR) is set, and the feedrate of the tool is set to F×(MDR) when Rc/Rp ≤ MDR.
When this parameter is 0, the minimum deceleration ratio (MDR) is 100%.

1711	Inner determination angle (θp) for inner corner override

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 2 to 178

Set an inner determination angle for inner corner override in automatic corner overriding.

1712	Override value for inner corner override

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 1 to 100
Set an inner corner override value in automatic corner overriding.

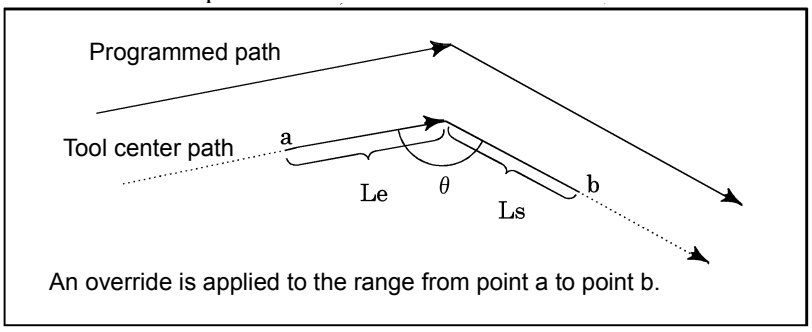
1713	Start distance (Le) for inner corner override

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set a start distance for inner corner override in automatic corner overriding.

1714	End distance (Ls) for inner corner override

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set an end distance for inner corner override in automatic corner overriding.

When $\theta \leq \theta p$, an inner corner is assumed. (Parameter No. 1711 is used to set θp.)
When a corner is determined to be an inner corner, an override is applied to the feedrate in the range of Le in the previous block from the intersection of the corner and in the range of Ls in the next block from the intersection of the corner.
Distances Le and Ls represent linear distances from the intersection of a corner to points on the tool center path.
Le and Ls are set in parameters Nos. 1713 and 1714.



1732	Minimum allowable feedrate for the deceleration function based on acceleration in circular interpolation

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis

[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
With the deceleration function based on acceleration in circular interpolation, an optimum feedrate is automatically calculated so that acceleration produced by changing the move direction in circular interpolation does not exceed the maximum allowable acceleration rate specified in parameter No. 1735.
If the radius of an arc is very small, a calculated feedrate may become too low.
In such a case, the feedrate is prevented from decreasing below the value specified in this parameter.

NOTE
During involute interpolation, the minimum allowable feedrate of "clamping of acceleration near a basic circle" in involute interpolation automatic feedrate control is used.

1735 **Maximum allowable acceleration rate for the deceleration function based on acceleration in circular interpolation for each axis**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, degree/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)
Set a maximum allowable acceleration rate for the deceleration function based on acceleration in circular interpolation.
Feedrate is controlled so that acceleration produced by changing the move direction in circular interpolation does not exceed the value specified in this parameter.
For an axis with 0 set in this parameter, the deceleration function based on acceleration is disabled.
If a different value is set in this parameter for each axis, a feedrate is determined from the smaller of the acceleration rates specified for the two circular axes.

NOTE
During involute interpolation, the minimum allowable feedrate of "clamping of acceleration near a basic circle" in involute interpolation automatic feedrate control is used.

1737 **Maximum allowable acceleration rate for the deceleration function based on acceleration in AI contour control for each axis**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, degree/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)
Set a maximum allowable acceleration rate produced by changing the tool move direction.
For an axis with 0 set in this parameter, the deceleration function based on acceleration is disabled. If 0 is set for all axes, the deceleration function based on acceleration is not performed.

In circular interpolation, however, the deceleration function based on feedrate control using acceleration in circular interpolation (parameter No. 1735) is enabled.

1738	Minimum allowable feedrate for the deceleration function based on acceleration in AI contour control

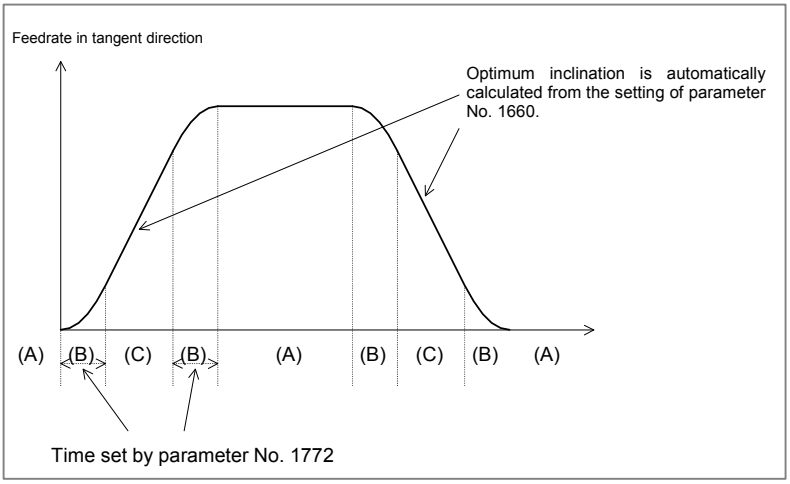
[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
With the deceleration function based on acceleration in AI contour control, a feedrate most suitable for a desired figure is automatically calculated.
Depending on the figure, however, the calculated feedrate may become too low.
In such a case, the feedrate is prevented from decreasing below the value specified in this parameter.
If overriding using the deceleration function based on cutting load is enabled, a feedrate lower than the minimum allowable feedrate may be used.

1769	Time constant for acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
In the acceleration/deceleration before interpolation mode as in AI contour control, not the ordinary time constant (parameter No. 1622) but the value of this parameter is used.
Be sure to specify the same time constant value for all axes except for a special application. If different values are set, correct linear and circular figures cannot be obtained.

1772	Acceleration change time of bell-shaped acceleration/deceleration before interpolation

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 200
Set an acceleration change time of bell-shaped acceleration/ deceleration before interpolation (time for changing from the state of constant feedrate (A) to the state of constant acceleration/deceleration (C) at the acceleration rate calculated from the acceleration rate set in parameter No. 1660: time of (B) in the figure).



1783 **Maximum allowable feedrate difference for feedrate determination based on corner feedrate difference**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
If a feedrate component change for each axis exceeding the value set in this parameter occurs at the joint of blocks, the feedrate determination function based on corner feedrate difference finds a feedrate not exceeding the set value and performs deceleration by using acceleration/deceleration before interpolation. Thus, a shock to the machine and machining error at a corner can be reduced.

1788 **Maximum allowable acceleration change rate in feedrate determination based on acceleration change for each axis**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, degree/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)
Set a maximum allowable acceleration change rate for each axis in feedrate control based on acceleration change under control on the rate of change of acceleration.
For an axis with 0 set in this parameter, feedrate control based on acceleration change is disabled.
If 0 is set for all axes, feedrate control based on acceleration change is not exercised.

1789 **Maximum allowable acceleration change rate in feedrate determination based on acceleration change for each axis (linear interpolation)**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, degree/sec^2 (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)

Set a maximum allowable acceleration change rate for each axis in feedrate control based on acceleration change under control on the rate of change of acceleration in successive linear interpolation operations.
In feedrate control based on acceleration change at a corner between linear interpolation operations, the maximum allowable acceleration change rate not set in parameter No. 1788 but set in this parameter is valid.
For an axis with 0 set in this parameter, the maximum allowable acceleration change rate set in parameter No. 1788 is valid.
Feedrate control based on acceleration change is disabled for an axis with 0 set in parameter No. 1788, so that the setting of this parameter for such an axis is ignored.

1790	**Ratio of change time of the rate of change of acceleration in smooth bell-shaped acceleration/deceleration before interpolation**

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 50

Set the ratio of the change time of the rate of change of acceleration to the change time of acceleration(*1) by percentage (%) in smooth bell-shaped acceleration/deceleration before look-ahead interpolation.
If 0 is set in this parameter or a value not within the valid data range is specified in this parameter, smooth bell-shaped acceleration/ deceleration before look-ahead interpolation is not performed.

(*1) Parameter No. 1772 for acceleration/deceleration before look-ahead interpolation (cutting feed).
Parameter No. 1672 for acceleration/deceleration before interpolation in linear rapid traverse, or for optimum torque acceleration/ deceleration.

	#7	#6	#5	#4	#3	#2	#1	#0
1802						DC2x	DC4x	

[Input type] Parameter input
[Data type] Bit axis

#1 DC4x When the reference position is established on the linear scale with reference marks:
0: An absolute position is established by detecting three reference marks.
1: An absolute position is established by detecting four reference marks.

#2 DC2x Reference position establishment operation for a linear scale with reference marks is performed as follows:
0: The setting of bit 1 (DC4) of parameter No. 1802 is followed.
1: An absolute position is established by detecting two reference marks.

NOTE
1 When this parameter is set to 1, specify the direction of the scale zero point by setting bit 4 (SCP) of parameter No. 1817.
2 When a rotary encoder with absolute address reference marks is used, this parameter is invalid. Even when this parameter is set to 1, the setting of bit 1 (DC4) of parameter No. 1802 is followed.

	#7	#6	#5	#4	#3	#2	#1	#0
1815			APCx	APZx	DCRx		OPTx	

[Input type] Parameter input
[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#1 OPTx Position detector
0: A separate Pulsecoder is not used.
1: A separate Pulsecoder is used.

NOTE
Set this parameter to 1 when using a linear scale with reference marks or a linear scale with an absolute address zero point (full-closed system).

#3 DCRx As a scale with absolute address reference marks:
0: A rotary encoder with absolute address reference marks is not used.
1: A rotary encoder with absolute address reference marks is used.

NOTE
When using a rotary encoder with absolute address reference marks, set also bit 2 (DCLx) of parameter No. 1815 to 1.

#4 APZx Machine position and position on absolute position detector when the absolute position detector is used

0: Not corresponding
1: Corresponding

When an absolute position detector is used, after primary adjustment is performed or after the absolute position detector is replaced, this parameter must be set to 0, power must be turned off and on, then manual reference position return must be performed. This completes the positional correspondence between the machine position and the position on the absolute position detector, and sets this parameter to 1 automatically.

#5 APCx Position detector

0: Other than absolute position detector
1: Absolute position detector (absolute Pulsecoder)

	#7	#6	#5	#4	#3	#2	#1	#0
1817		TANx						

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#6 TANx Tandem control

0: Not used
1: Used

NOTE
Set this parameter to both master axis and slave axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1818					SDC		RF2x	RFSx

[Input type] Parameter input
[Data type] Bit axis

#0 RFSx If G28 is specified for an axis for which a reference position is not established (ZRF <Fn120>= "0") when a linear scale with an absolute address zero point or a linear scale with absolute address reference marks is used:

0: A movement is made to the reference position after reference position establishment operation.
1: No movement is made after reference position establishment operation, but the operation is completed.

NOTE
This parameter disables movement based on the G28 command to a reference position. So, use this parameter only in special cases.

#1 RF2x If G28 is specified for an axis for which a reference position is not established (reference position establishment signal ZRF = 0) when a linear scale with an absolute address zero point or a linear scale with absolute address reference marks is used:

0: A movement is made to the reference position.
1: No movement is made to the intermediate position and reference position, but the operation is completed.

NOTE
This parameter disables movement based on the G28 command to a reference position. So, use this parameter only in special cases.

#3 SDCx A linear scale with an absolute address zero point is:

0: Not used.
1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
1819						DATx		

[Input type] Parameter input
[Data type] Bit axis

#2 DATx When a linear scale with an absolute address zero point or a linear scale with absolute address reference marks is used, the automatic setting of parameters Nos. 1883 and 1884 at manual reference position return time is:

0: Not performed.
1: Performed.

The automatic setting procedure is as follows:

<1> Set an appropriate value in parameters Nos. 1815, 1821, and 1882.
<2> Position the machine at the reference position by manual operation.
<3> Set this parameter to 1.
<4> Perform a manual reference position return operation. Upon completion of manual reference position return operation, parameters Nos. 1883 and 1884 are set, and this parameter is automatically set to 0.

1820	Command multiplier for each axis (CMR)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] See below :

Set a command multiplier indicating the ratio of the least command increment to the detection unit for each axis.
Least command increment = detection unit × command multiplier

Relationship between the increment system and the least command increment

(1) T series

			Least input increment	Least command increment
IS-B	Millimeter machine	Millimeter input	0.001 mm (diameter specification)	0.0005 mm
			0.001 mm (radius specification)	0.001 mm
		Inch input	0.0001 inch (diameter specification)	0.0005 mm
			0.0001 inch (radius specification)	0.001 mm
	Inch machine	Millimeter input	0.001 mm (diameter specification)	0.00005 inch
			0.001 mm (radius specification)	0.0001 inch
		Inch input	0.0001 inch (diameter specification)	0.00005 inch
			0.0001 inch (radius specification)	0.0001 inch
	Rotary axis		0.001 deg	0.001 deg

			Least input increment	Least command increment
IS-C	Millimeter machine	Millimeter input	0.0001 mm (diameter specification)	0.00005 mm
			0.0001 mm (radius specification)	0.0001 mm
		Inch input	0.00001 inch (diameter specification)	0.00005 mm
			0.00001 inch (radius specification)	0.0001 mm
	Inch machine	Millimeter input	0.0001 mm (diameter specification)	0.000005 inch
			0.0001 mm (radius specification)	0.00001 inch
		Inch input	0.00001 inch (diameter specification)	0.000005 inch
			0.00001 inch (radius specification)	0.00001 inch
	Rotary axis		0.0001 deg	0.0001 deg

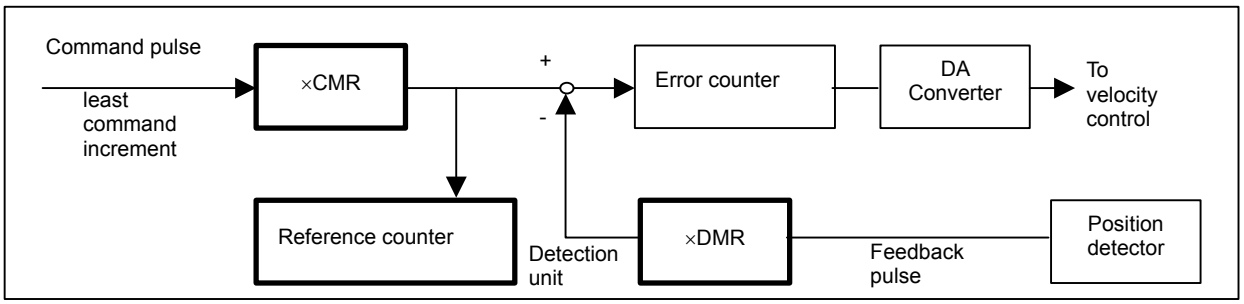
			Least input increment	Least command increment
IS-D	Millimeter machine	Millimeter input	0.00001 mm (diameter specification)	0.000005 mm
			0.00001 mm (radius specification)	0.00001 mm
		Inch input	0.000001 inch (diameter specification)	0.000005 mm
			0.000001 inch (radius specification)	0.00001 mm
	Inch machine	Millimeter input	0.00001 mm (diameter specification)	0.0000005 inch
			0.00001 mm (radius specification)	0.000001 inch
		Inch input	0.000001 inch (diameter specification)	0.0000005 inch
			0.000001 inch (radius specification)	0.000001 inch
	Rotary axis		0.00001 deg	0.00001 deg

			Least input increment	Least command increment
IS-E	Millimeter machine	Millimeter input	0.000001 mm (diameter specification)	0.0000005 mm
			0.000001 mm (radius specification)	0.000001 mm
		Inch input	0.0000001 inch (diameter specification)	0.0000005 mm
			0.0000001 inch (radius specification)	0.000001 mm
	Inch machine	Millimeter input	0.000001 mm (diameter specification)	0.00000005 inch
			0.000001 mm (radius specification)	0.0000001 inch
		Inch input	0.0000001 inch (diameter specification)	0.00000005 inch
			0.0000001 inch (radius specification)	0.0000001 inch
	Rotary axis		0.000001 deg	0.000001 deg

(2) M series

Increment system	Least input increment and least command increment					
	IS-A	IS-B	IS-C	IS-D	IS-E	Unit
Millimeter machine	0.01	0.001	0.0001	0.00001	0.000001	mm
Millimeter input	0.001	0.0001	0.00001	0.000001	0.0000001	inch
Rotary axis	0.01	0.001	0.0001	0.00001	0.000001	deg

Setting command multiply (CMR), detection multiply (DMR), and the capacity of the reference counter



Set CMR and DMR so that the pulse weight of + input (command from the CNC) into the error counter matches the pulse weight of -input (feedback from the position detector).

[Least command increment]/CMR=[Detection unit]=
[Feedback pulse unit]/DMR

[Least command increment]
Minimum unit of commands issued from the CNC to the machine

[Detection unit]
Minimum unit for machine position detection

The unit of feedback pulses varies, depending on the type of detector.

[Feedback pulse unit]=
[Amount of travel per rotation of the Pulsecoder]/
[Number of pulses per rotation of the Pulsecoder]

As the size of the reference counter, specify the grid interval for the reference position return in the grid method.

[Size of the reference counter]=[Grid interval]/[Detection unit]
[Grid interval]=[Amount of travel per rotation of the Pulsecoder]

The setting of a command multiplier is as follows:

(1) When command multiplier is 1 to 1/27
Set value = 1 / command multiplier + 100
Valid data range : 101 to 127

(2) When command multiply is 0.5 to 48
Set value = 2 × command multiplier
Valid data range : 1 to 96

NOTE
If a feedrate exceeding the feedrate found by the expression below is used, an incorrect travel amount may result or a servo alarm may be issued. Be sure to use a feedrate not exceeding the feedrate found by the following expression:
Fmax[mm/min] = 196602 × 10^4 × least command increment / CMR

1821	Reference counter size for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis

[Unit of data] Detection unit
[Valid data range] 0 to 999999999
Set a reference counter size.
As a reference counter size, specify a grid interval for reference position return based on the grid method.
When a value less than 0 is set, the specification of 10000 is assumed.
When a linear scale with absolute address reference marks is used, set the interval of mark 1.

1828	Positioning deviation limit for each axis in movement

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
Set the positioning deviation limit in movement for each axis.
If the positioning deviation exceeds the positioning deviation limit during movement, a servo alarm SV0411, "EXCESS ERROR (MOVING)" is generated, and operation is stopped immediately (as in emergency stop).
Generally, set the positioning deviation for rapid traverse plus some margin in this parameter.

1829	Positioning deviation limit for each axis in the stopped state

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
Set the positioning deviation limit in the stopped state for each axis.
If, in the stopped state, the positioning deviation exceeds the positioning deviation limit set for stopped state, a servo alarm SV0410, "EXCESS ERROR (STOP)" is generated, and operation is stopped immediately (as in emergency stop).

1838	Position deviation limit for each axis in moving during safety check

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
Position deviation limit for each axis during moving for safety check of Dual Check Safety function is specified.
If position deviation of a moving axis exceeds position deviation limit while Safety Check is carried out (Safety Monitoring Request "*VLDVx" =0), a servo alarm SV0475, "EXCESS ERROR(MOVE:SV)" or SV1071, "EXCESS ERROR(MOVE:CNC)" is generated and axes are stopped immediately like emergency stop state..
In Dual Check Safety function, position deviation is always checked by CNC and Servo. In case that Safety Check is carried out (Safety Monitoring Request "*VLDVx" =0), the servo alarm SV0475 or SV1071 is generated when each CPU finds out that the deviation exceeds position deviation limit in moving state.

1841	Position deviation limit of each axis in moving state during other than Dual Check Safety monitoring (for Dual Check Safety Function)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2 word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

Set the positioning deviation limit in moving state for each axis for Dual Check Safety function, in case that Safety Check is not carried out (Safety Monitoring Request "*VLDVx"=1).
In case that Safety Check is not carried out (Safety Monitoring Request "*VLDVx" =1), servo alarm SV0475, "EXCESS ERROR(MOVE:SV)" or SV1071, "EXCESS ERROR(MOVE:CNC)" is generated and operation is stopped immediately (as in emergency stop), when each CPU finds out that the deviation exceeds position deviation limit in moving state.
If the value of this parameter is 0, the parameter No.1828 is used for the value of deviation limit in moving state.
In case that Safety Check is carried out (Safety Monitoring Request "*VLDVx" =0), the parameter No.1838 is used for the value of deviation limit in moving state.

1851	Backlash compensating value for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] -9999 to 9999

Set the backlash compensating value for each axis.
When the machine moves in a direction opposite to the reference position return direction after the power is turned on, the first backlash compensation is performed.

1882	Interval of mark 2 of a linear scale with absolute address reference marks

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

Set the interval of mark 2 of a linear scale with absolute address reference marks.

1883	Distance 1 from the scale zero point to reference position (linear scale with absolute address reference marks) or distance 1 from the base point to reference position (linear scale with an absolute address zero point)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999999999 to 999999999

1884	**Distance 2 from the scale zero point to reference position (linear scale with absolute address reference marks) or distance 2 from the base point to reference position (linear scale with an absolute address zero point)**

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999 to 999

When a linear scale with absolute address reference marks is used, set the distance from the scale zero point to reference position in parameter Nos. 1883 and 1884).

Distance from the zero point to the reference position of a linear scale
= No. 1884 × 1,000,000,000 + No. 1883

The scale zero point represents a point where mark 1 and mark 2 match. Usually, this point is a virtual point that does not physically exist on the scale. (See the Fig. A.1 (a).)
If the reference position is placed in the + direction when viewed from the scale zero point, set a positive value. If the reference position is placed in the - direction when viewed from the scale zero point, set a negative value.

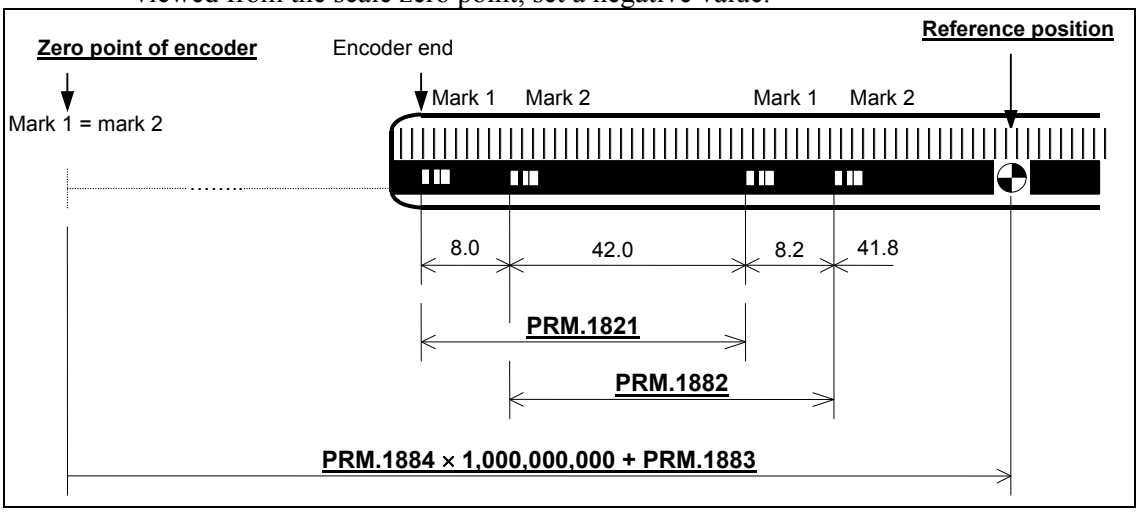
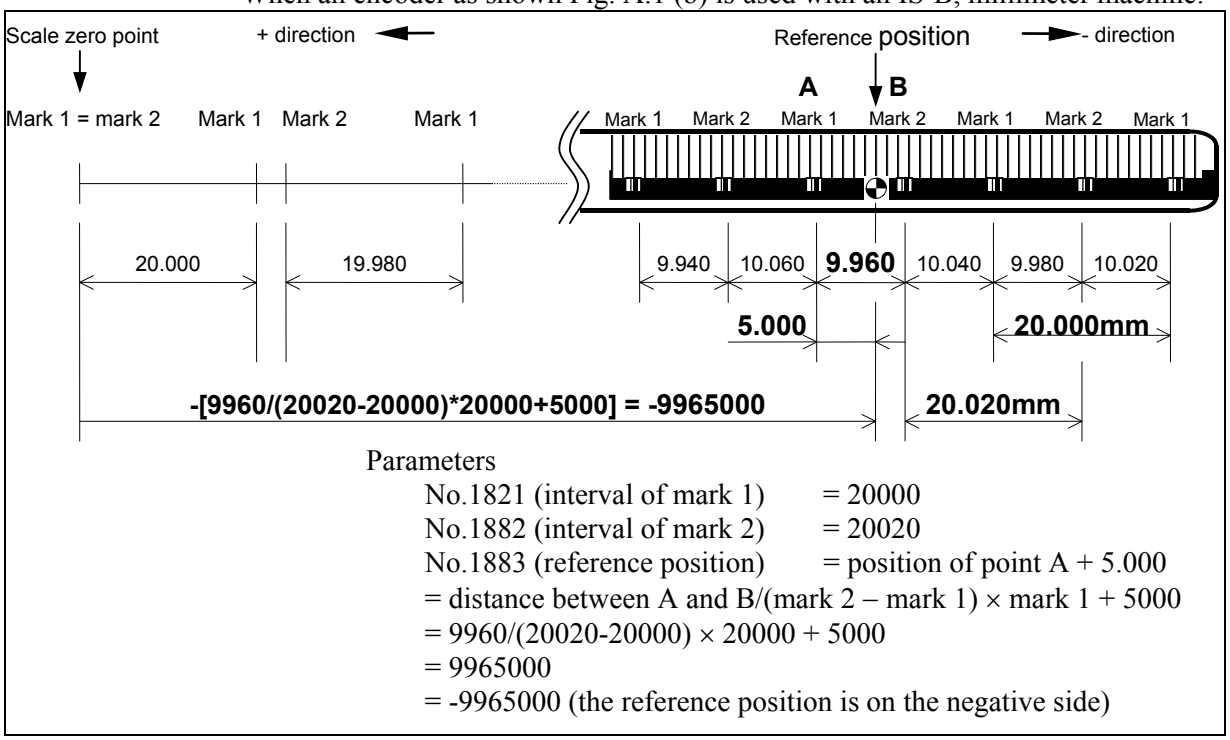


Fig. A.1 (a)

[Example of parameter settings]

When an encoder as shown Fig. A.1 (b) is used with an IS-B, millimeter machine:



Parameters

No.1821 (interval of mark 1) = 20000
No.1882 (interval of mark 2) = 20020
No.1883 (reference position) = position of point A + 5.000
= distance between A and B/(mark 2 – mark 1) × mark 1 + 5000
= 9960/(20020-20000) × 20000 + 5000
= 9965000
= -9965000 (the reference position is on the negative side)

Fig. A.1 (b)

[Setting parameter No. 1883]

When it is difficult to measure the distance from the scale zero point to the reference position (parameter No. 1883), the method described below can be used to find the distance.

<1> Set parameter No. 1815 to enable this function.
Set an appropriate value in parameter No. 1821 and No. 1882.
Set 0 in parameter No. 1240.
Set 0 in parameter No. 1883 and No. 1884.
<2> At an appropriate position, establish a reference position.
(As a result, the machine coordinate represents the distance from the scale zero point to the current position.)
<3> By jog feed or handle feed, place the machine at the accurate reference position.
<4> In parameter No. 1883, set the machine coordinate of that time converted to the detection unit (machine coordinate × CMR).
<5> If necessary, set parameter No. 1240.

When a linear scale with an absolute address zero point is used, set the distance from the base point to the reference position in parameter Nos. 1883 and 1884. The base point is a point at a scale end as shown Fig. A.1 (c).

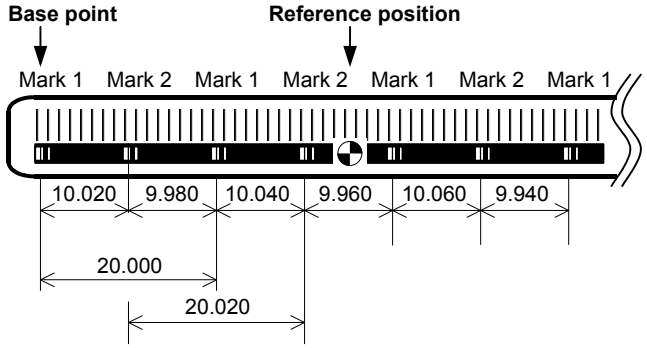


Fig. A.1 (c)

If the reference position is located in the positive direction when viewed from the base point, set a positive value; if the reference position is located in the negative direction, set a negative value. Set the value by following the steps explained below.

<1> Set bit 1 (OPT) of parameter No. 1815 , bit 2 (DCL) of parameter No. 1815, and bit 3 (SDC) of parameter No. 1818 to enable this function.
Set 0 in parameter No. 1240.
Set 0 in parameter Nos. 1883 and 1884.

<2> At an appropriate position, establish a reference position.
(Consequently, the machine coordinate value indicates the distance from the base point to current position.)

<3> By jog feed or handle feed, place the machine at the accurate reference position.

<4> In parameters Nos. 1883 and 1884, set the machine coordinate of that time converted to the detection unit (machine coordinate × CMR).
If necessary, set parameter No. 1240.

NOTE
1 Set parameter Nos. 1883 and 1884 so that the distance from the scale zero point (for a linear scale with absolute address reference marks) or the base point (for a linear scale with an absolute address zero point) to the reference position is within the range from -999,999,999,999 to +999,999,999,999. If a value beyond this range is set, an alarm DS0016 or DS1448 is issued.
2 The scale area on the scale cannot be extended across the scale zero point or base point. Make parameter settings not to cause the scale area to extend beyond the scale zero point or base point.

	#7	#6	#5	#4	#3	#2	#1	#0
1902							ASE	FMD

[Input type] Parameter input
[Data type] Bit

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 FMD The FSSB setting mode is:
0: Automatic setting mode.
(When the relationship between an axis and amplifier is defined on the FSSB setting screen, parameters Nos. 1023, 2013#0, 2014#0, 3717, 11802#4, 24000 to 24103 are automatically set.
1: Manual setting 2 mode.
(Parameters Nos. 1023, 2013#0, 2014#0, 3717, 11802#4, 24000 to 24103 are to be manually set.)

#1 ASE When automatic setting mode is selected for FSSB setting (when the bit 0 (FMD) parameter No. 1902 is set to 0), automatic setting is:
0: Not completed.
1: Completed.
This bit is automatically set to 1 upon the completion of automatic setting.

Parameters Nos. 2000 to 2999 are for digital servo, The following parameters are not explained in this manual. Refer to FANUC AC SERVO MOTOR αi series PARAMETER MANUAL (B-65270EN)

	#7	#6	#5	#4	#3	#2	#1	#0
2011	XIAx							SYNx

[Input type] Parameter input
[Data type] Bit axis

#0 SYNx When the electronic gear box function (EGB) is used, this bit sets the axis to be synchronized.
0: Axis not synchronized by EGB
1: Axis synchronized by EGB
Set 1 for both of the slave and dummy axes of EGB.

NOTE
The setting of this parameter becomes valid after the power is turned off then back on.

#7 XIAx Temporary absolute coordinate setting is:
0: Not used.
1: Used.

NOTE
1 When temporary absolute coordinate setting is used, bit 1 (OPTx) of parameter No. 1815, bit 5 (APCx) of parameter No. 1815, parameter No. 1874, and parameter No. 1875 must be set.
2 The setting of this parameter becomes effective after the power is turned off then back on.

2031	Torque command difference threshold of torque difference alarm

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 14564
If the absolute value of the torque command difference between two axes exceeds the value set in this parameter, an alarm is issued.

Set the same value for two axes that are placed under axis synchronous control.
The servo axis numbers of the synchronized master axis and slave axis must be assigned so that an odd number is assigned to the master axis and the next axis number is assigned to the slave axis. Examples are (1,2) and (3,4).

	#7	#6	#5	#4	#3	#2	#1	#0
3002	OVM	POV						

[Input type] Parameter input
[Data type] Bit path

#6 POV Dwell/Auxiliary function time override function is:
0: Invalid.
1: Valid.

#7 OVM In Dwell/Auxiliary function time override function, override function for M02,M30 is:
0: Invalid.
1: Valid.

	#7	#6	#5	#4	#3	#2	#1	#0
3008						XSG		

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit path

#2 XSG A signal assigned to an X address is:
0: Fixed at the address.
1: Able to be reassigned to an arbitrary X address.

NOTE
When this parameter is set to 1, set parameter No. 3013, No. 3014, No. 3012, and No. 3019. If parameter No. 3013 and No. 3014 are not set, the deceleration signal for reference position return is assigned to bit 0 of <X0000>. If parameter No. 3012 and No. 3019 are not set, the skip signal, the PMC axis control skip signal, the measurement position arrival signal, the manual feed interlock signal for each axis direction, and the tool compensation value write signal are assigned to <X0000>.

3012	Skip signal assignment address

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 727
Set an X address to which the skip signal SKIPn is to be assigned.

NOTE
This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
Depending on the configuration of the I/O Link, the actually usable X addresses are:
<X0000 to X0127>, <X0200 to X0327>, <X0400 to X0527>, <X0600 to X0727>

3013	X address to which the deceleration signal for reference position return is assigned

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 727
Set an address to which the deceleration signal *DECn for reference position return for each axis is to be assigned.

NOTE
This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
Depending on the configuration of the I/O Link, the actually usable X addresses are:
<X0000 to X0127>, <X0200 to X0327>, <X0400 to X0527>, <X0600 to X0727>

3018	Percentage assumed when the 1% rapid traverse override signal indicates 0% during execution of an auxiliary function

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 100
When the 1% rapid traverse override signal indicates 0% in the dwell/auxiliary function time override function, this parameter sets the percentage for calculating an insufficient time.
0% is assumed to be 10%.

3019	Address to which the PMC axis control skip signal, measurement position arrival signal, and tool offset write signals are assigned

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 727

Set an X address to which the PMC axis control skip signal ESKIP, measurement position arrival signals (XAE, YAE, and ZAE (M series) or XAE and ZAE (T series)), and tool offset write signals (±MIT1 and ±MIT2 (T series)) are to be assigned.

Example 1. When parameter No. 3012 is set to 5 and parameter No. 3019 is set to 6

When bit 2 (XSG) of parameter No. 3008 is 1, the PMC axis control skip signal, and measurement position arrival signal are allocated to X0006 and the skip signal is allocated to X0005.

	#7	#6	#5	#4	#3	#2	#1	#0	
X0005	SKIP	SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	(T series)
	#7	#6	#5	#4	#3	#2	#1	#0	
	SKIP	SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	(M series)

	#7	#6	#5	#4	#3	#2	#1	#0	
X0006		ESKIP	-MIT2	+MIT2	-MIT1	+MIT1	ZAE	XAE	(T series)
	#7	#6	#5	#4	#3	#2	#1	#0	
		ESKIP				ZAE	YAE	XAE	(M series)

Example 2. When parameter No. 3012 is set to 5 and parameter No. 3019 is set to 5

When bit 2 (XSG) of parameter No. 3008 is 1, the PMC axis control skip signal, measurement position arrival signal, and skip signal are allocated to X0005.

	#7	#6	#5	#4	#3	#2	#1	#0	
X0005	SKIP	ESKIP	-MIT2	+MIT2	-MIT1	+MIT1	ZAE	XAE	(T series)
		SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	
	#7	#6	#5	#4	#3	#2	#1	#0	
	SKIP	ESKIP	SKIP5	SKIP4	SKIP3	ZAE	YAE	XAE	(M series)
		SKIP6				SKIP2	SKIP8	SKIP7	

NOTE
This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.
Depending on the configuration of the I/O Link, the actually usable X addresses are:
<X0000 to X0127>, <X0200 to X0327>, <X0400 to X0527>, <X0600 to X0727>

3021	Address to which an axis signal is assigned

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7, 10 to 17, 20 to 27, ... , 90 to 97

For each axis of the CNC, set a PMC interface address.
Set a value according to the tables below.

Value of parameter No. 3021 (the second digit)

Setting value	Input signal address	Output signal address
0	G0000 to G0767	F0000 to F0767
1	G1000 to G1767	F1000 to F1767
	:	
9	G9000 to G9767	F9000 to F9767

Value of parameter No. 3021 (the first digit)

Setting value	Input signal address	Output signal address
0	0	0
1	1	1
	:	
7	7	7

[Example of setting]

Axis number	No. 3021	Signal allocation
1	0	+J1<G0100.0>, -J1<G0102.0>, ZP1<F0090.0>, ...
2	1	+J2<G0100.1>, -J2<G0102.1>, ZP2<F0090.1>, ...
3	2	+J3<G0100.2>, -J3<G0102.2>, ZP3<F0090.2>, ...
4	10	+J4<G1100.0>, -J4<G1102.0>, ZP4<F1090.0>, ...
5	11	+J5<G1100.1>, -J5<G1102.1>, ZP5<F1090.1>, ...

If eight or less axes are used per path, the following signal allocation results when 0 is set for all axes:

Axis 1 of path 1 = Setting equivalent to 0
Axis 2 of path 1 = Setting equivalent to 1
:
Axis 1 of path 2 = Setting equivalent to 10
:

NOTE
Set this parameter when more than eight axes are used per path. The valid data range varies, depending on the system software.

3022 **Address to which a spindle signal is assigned**

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 3, 10 to 13, 20 to 23, ... , 90 to 93

For each axis of the CNC, set a PMC interface address.
Set a value according to the tables below.

Value of parameter No. 3022 (the second digit)

Setting value	Input signal address	Output signal address
0	G0000toG0767	F0000toF0767
1	G1000toG1767	F1000toF1767
	:	
9	G9000toG9767	F9000toF9767

Value of parameter No. 3022 (the first digit)

Setting value	Input signal address	Output signal address
0	Bit position A	Bit position A
1	Bit position B	Bit position B
2	Bit position C	Bit position C
3	Bit position D	Bit position D

(The bit positions A, B, C, and D vary, depending on the type of signal.)

[Example of setting]

Spindle number	No. 3022	Signal allocation
1	0	TLMLA<G0070.0>, TLMHA<G0070.1>, ALMA<F0045.0>, ...
2	1	TLMLB<G0074.0>, TLMHB<G0074.1>, ALMB<F0049.0>, ...
3	10	TLMLA<G1070.0>, TLMHA<G1070.1>, ALMA<F1045.0>, ...
4	11	TLMLB<G1074.0>, TLMHB<G1074.1>, ALMB<F1049.0>, ...

If four or less axes are used per path, the following signal allocation results when 0 is set for all axes:

Axis 1 of path 1 = Setting equivalent to 0
Axis 2 of path 1 = Setting equivalent to 1
:
Axis 1 of path 2 = Setting equivalent to 10
:

NOTE
The valid data range varies, depending on the system software.

3030	Allowable number of digits for the M code

3031	Allowable number of digits for the S code

3032	Allowable number of digits for the T code

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 8
Set the allowable numbers of digits for the M, S, and T codes.
When 0 is set, the allowable number of digits is assumed to be 8.

3033	Allowable number of digits for the B code (second auxiliary function)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 8
Set the allowable number of digits for the second auxiliary function.
When 0 is set, the allowable number of digits is assumed to be 8.
To enable a decimal point to be specified, bit 0 (AUP) of parameter No. 3450 must be set to 1. In this case, the allowable number of digits set in this parameter includes the number of decimal places.
If a value exceeding the allowable number of digits is specified, the alarm PS0003, “TOO MANY DIGIT” is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
3104	DAC		DRC		PPD			MCN
	DAC	DAL	DRC	DRL	PPD			MCN

[Input type] Parameter input
[Data type] Bit path

#0 MCN Machine position
0: Regardless of whether input is made in mm or inches, the machine position is displayed in mm for millimeter machines, or in inches for inch machines.
1: When input is made in mm, the machine position is displayed in mm, and when input is made in inches, the machine position is displayed in inches accordingly.

#3 PPD Relative position display when a coordinate system is set
0: Not preset
1: Preset

> **NOTE**
> If any of the following is executed when PPD is set to 1, the relative position display is preset to the same value as the absolute position display:
> (1) Manual reference position return
> (2) Coordinate system setting based on G92 (G50 for G code system A on the lathe system)
> (3) Workpiece coordinate system presetting based on G92.1 (G50.3 for G code system A on the lathe system)
> (4) When a T code for the lathe system is specified.

#4 DRL Relative position
0: The actual position displayed takes into account tool length offset.
1: The programmed position displayed does not take into account tool length offset.

#5 DRC When relative positions are displayed:
0: Values not excluding the amount of travel based on cutter compensation and tool nose radius compensation are displayed.
1: Values excluding the amount of travel based on cutter compensation and tool nose radius compensation (programmed positions) are displayed.

#6 DAL Absolute position
0: The actual position displayed takes into account tool length offset.
1: The programmed position displayed does not take into account tool length offset.

> **NOTE**
> In lathe systems, whether to exclude a tool offset when displaying the absolute position is determined by the setting of bit 1 (DAP) of parameter No. 3129.

#7 DAC When an absolute position are displayed:
0: Values not excluding the amount of travel based on cutter compensation and tool nose radius compensation are displayed.
1: Values excluding the amount of travel based on cutter compensation and tool nose radius compensation (programmed positions) are displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3106		DAK						

[Input type] Setting input
[Data type] Bit

#6 DAK Specifies whether to display coordinates in the program coordinate system or workpiece coordinate system as absolute coordinates when the 3-dimensional coordinate conversion mode, the tilted working plane indexing mode or the workpiece setting error compensation mode is set.
0: Display coordinates in the program coordinate system.
1: Display coordinates in the workpiece coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
3107	MDL			SOR				

[Input type] Setting input
[Data type] Bit path

#4 SOR Display of the program directory
0: Programs are listed in the order of registration.
1: Programs are listed in the order of name.

NOTE
In the file list on the data server, the programs are displayed in the order of program number with zeros suppressed if the parameter is 0.

#7 MDL Display of the modal state on the program edit screen in 8.4-inch display unit
0: Not displayed.
1: Displayed (only in the MDI mode).

	#7	#6	#5	#4	#3	#2	#1	#0
3111		OPS	OPM				SPS	SVS

[Input type] Setting input
[Data type] Bit path

#0 SVS Servo setting screen and servo tuning screen
0: Not displayed
1: Displayed

#1 SPS Spindle tuning screen
0: Not displayed
1: Displayed

#5 OPM Operating monitor
0: Not displayed
1: Displayed

#6 OPS The speedometer on the operating monitor screen indicates:
0: Spindle motor speed
1: Spindle speed

	#7	#6	#5	#4	#3	#2	#1	#0
3115							NDAx	NDPx

[Input type] Parameter input
[Data type] Bit axis

#0 NDPx The current position is:
0: Displayed.
1: Not displayed.

NOTE
When using the electronic gear box (EGB) function, set 1 for the EGB dummy axis to disable current position display.

#1 NDAx The current position and the amount of the movement to be made in absolute and relative coordinates are:
0: Displayed.
1: Not displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3129							DAP	DRP

[Input type] Parameter input
[Data type] Bit path

#0 DRP For relative coordinate display:
0: The actual position considering a tool offset (tool movement) is displayed.
1: The programmed position excluding a tool offset (tool movement) is displayed.

#1 DAP For absolute coordinate display:
0: The actual position considering a tool offset (tool movement) is displayed.
1: The programmed position excluding a tool offset (tool movement) is displayed.

NOTE
In machining center systems, whether to exclude the tool length offset when displaying the absolute position is determined according to the setting of bit 6 (DAL) of parameter No. 3104.

3131	Subscript of axis name

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 9, 65 to 90
In order to distinguish axes under parallel operation, synchronization control, and tandem control, specify a subscript for each axis name.

Setting value	Meaning
0	Each axis is set as an axis other than a parallel axis, synchronization control axis, and tandem control axis.
1 to 9	A set value is used as a subscript.
65 to 90	A set letter (ASCII code) is used as a subscript.

[Example] When the axis name is X, a subscript is added as indicated below.

Setting value	Axis name displayed on a screen such as the position display screen
0	X
1	X1
77	XM
83	XS

If a multi-path system is used, no extended axis name is used within a path, and no subscript is set for the axis names, then the path number is automatically used as the subscript for the axis names. To disable the display of axis name subscripts, set a blank (32) of ASCII code in the parameter for specifying an axis name subscript.

NOTE
If even one axis in a path uses an extended axis name when bit 2 (EAS) of parameter No. 11308 is set to 0, subscripts cannot be used for axis names in the path.

3141	Path name (1st character)
3142	Path name (2nd character)
3143	Path name (3rd character)
3144	Path name (4th character)
3145	Path name (5th character)
3146	Path name (6th character)
3147	Path name (7th character)

[Input type] Parameter input
[Data type] Word path
[Valid data range] See the character-code correspondence table.
Specify a path name with codes.
Any character string consisting of alphanumeric characters, katakana characters, and special characters with a maximum length of seven characters can be displayed as a series name.

NOTE
1 For characters and codes, see Appendix A, “CHARACTER CODE LIST”.
2 When 0 is set in parameter No. 3141, PATH1(,PATH2...) are displayed as path names.
3 When optional path name display is enlarged (with bit 2 (PNE) of parameter No. 11350 set to 1), only alphanumeric characters are displayed. If any other type of characters are set, spaces are displayed instead.

	#7	#6	#5	#4	#3	#2	#1	#0
3201		NPE						

[Input type] Parameter input
[Data type] Bit path

#6 NPE With an M02, M30, or M99 block, program registration is assumed to be:
0: Completed
1: Not completed

	#7	#6	#5	#4	#3	#2	#1	#0
3202				NE9				NE8

[Input type] Parameter input
[Data type] Bit path

#0 NE8 Editing of subprograms with program numbers 8000 to 8999

0: Not inhibited
1: Inhibited

When this parameter is set to 1, the following editing operations are disabled:

(1) Program deletion (Even when deletion of all programs is specified, programs with program numbers 8000 to 8999 are not deleted.)
(2) Program output (Even when outputting all programs is specified, programs with program numbers 8000 to 8999 are not output.)
(3) Program number search
(4) Program editing of registered programs
(5) Program registration
(6) Program collation
(7) Displaying programs

NOTE
This parameter setting does not affect the following programs:
(1) Programs on the Data Server
(2) Programs for running and editing memory card programs on a memory card

#4 NE9 Editing of subprograms with program numbers 9000 to 9999

0: Not inhibited
1: Inhibited

When this parameter is set to 1, the following editing operations are disabled:

(1) Program deletion (Even when deletion of all programs is specified, programs with program numbers 9000 to 9999 are not deleted.)
(2) Program output (Even when outputting all programs is specified, programs with program numbers 9000 to 9999 are not output.)
(3) Program number search
(4) Program editing of registered programs
(5) Program registration
(6) Program collation
(7) Displaying programs

NOTE
This parameter setting does not affect the following programs:
(1) Programs on the Data Server
(2) Programs for running and editing memory card programs on a memory card

	#7	#6	#5	#4	#3	#2	#1	#0
3203	MCL	MER	MZE					

[Input type] Parameter input
[Data type] Bit path

#5 MZE After MDI operation is started, program editing during operation is:

0: Enabled
1: Disabled

#6 MER When the last block of a program has been executed at single block operation in the MDI mode, the executed block is:
0: Not deleted
1: Deleted

> **NOTE**
> When MER is set to 0, the program is deleted if the end-of-record mark (%) is read and executed. (The mark % is automatically inserted at the end of a program.)

#7 MCL Whether a program prepared in the MDI mode is cleared by reset
0: Not deleted
1: Deleted

	#7	#6	#5	#4	#3	#2	#1	#0
3204		MKP	SPR	P9E	P8E			

[Input type] Parameter input
[Data type] Bit path

#3 P8E Editing of subprograms with program numbers 80000000 to 89999999
0: Not inhibited.
1: Inhibited.

#4 P9E Editing of subprograms with program numbers 90000000 to 99999999
0: Not inhibited.
1: Inhibited.

#5 SPR A particular program number in the nine thousands is
0: Not assumed to be a number obtained by adding 90000000.
1: Assumed to be a number obtained by adding 90000000.

#6 MKP When M02, M30, or EOR(%) is executed during MDI operation, the created MDI program is:
0: Erased automatically.
1: Not erased automatically.

> **NOTE**
> If the bit 6 (MER) of parameter No. 3203 is set to 1, executing the last block provides a choice of whether to automatically erase a created program.

	#7	#6	#5	#4	#3	#2	#1	#0
3207			VRN					

[Input type] Parameter input
[Data type] Bit

#5 VRN On the custom macro variable screen, the variable names of common variables #500 to #549 are:
0: Not displayed.
1: Displayed.

3210	Program protection (PSW)

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 to 99999999

This parameter sets a password for protecting program Nos. 9000 to 9999. When a value other than zero is set in this parameter and this value differs from the keyword set in parameter No. 3211, bit 4 (NE9) of parameter No. 3202 for protecting program Nos. 9000 to 9999 is automatically set to 1.
This disables the editing of program Nos. 9000 to 9999. Until the value set as the password is set as a keyword, NE9 cannot be set to 0 and the password cannot be modified.

NOTE
1 The state where password ≠ 0 and password ≠ keyword is referred to as the locked state. When an attempt is made to modify the password by MDI input operation in this state, the warning message "WRITE PROTECTED" is displayed to indicate that the password cannot be modified. When an attempt is made to modify the password with G10 (programmable parameter input), alarm PS0231, “ILLEGAL FORMAT IN G10 L52” is issued.
2 When the value of the password is not 0, the parameter screen does not display the password. Care must be taken in setting a password.

3211	Program protection key (KEY)

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 to 99999999

When the value set as the password (set in parameter No. 3210) is set in this parameter, the locked state is released and the user can now modify the password and the value set in bit 4 (NE9) of parameter No. 3202.

NOTE
The value set in this parameter is not displayed. When the power is turned off, this parameter is set to 0.

3220	Password (PSW)

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 to 99999999

This parameter sets a password (PSW). When a value other than 0 is set, a password is set. When a password is set, a blank is displayed in this parameter, and the state (locked state) where an operation such as program editing is locked is set. When password (PSW) = 0, namely, in the normal state, or when password (PSW) = keyword (KEY), namely, in the unlock state, this parameter can be set.

3221	Keyword (KEY)

[Input type] Parameter input
[Data type] 2-word

[Valid data range] 0 to 99999999
When the same value as the password (PSW) is set in this parameter, the lock is released (unlock state). The value set in this parameter is not displayed.
The value of this parameter is initialized to 0 automatically when the power is turned on. So, if the power is turned off in the unlock state then is turned on again, the lock state is automatically set.

3222	Program protection range (minimum value) (PMIN)
3223	Program protection range (maximum value) (PMAX)

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 to 99999999
The programs in a range set here can be locked. Set the minimum program number and maximum program number of a desired range.
Set these parameters to satisfy PMAX > PMIN.
These parameters can be set when password (PSW) = 0, namely, in the normal state, or when password (PSW) = keyword (KEY), namely in the unlock state.
[Example] Parameter No. 3222 = 7000
Parameter No. 3223 = 8499
When the values above are set, the programs from O7000 to O8499 can be locked.
When PMIN = 0, the specification of PMIN = 9000 is assumed. When PMAX = 0, the specification of PMAX = 9999 is assumed. So, when these parameters are set to the defaults, the programs from O9000 to O9999 are locked.

NOTE
1 Parameters Nos. 3220 to 3223 are neither outputted nor input.
2 Parameters Nos. 3220 to 3223 are not cleared even when a parameter file clear operation is performed in the IPL state.
3 The values of a password (PSW) and keyword (KEY) are not displayed. When password (PSW) = 0, 0 is displayed in parameter No. 3220 to indicate that the normal state is set.
4 When a password (PSW) or keyword (KEY) is set, [+INPUT] has the same effect as [INPUT]. For example, if the input operation "1[+INPUT]" is performed when 99 is set in the keyword (KEY) parameter, 1 is set.

	#7	#6	#5	#4	#3	#2	#1	#0
3233							PDM	PCE

[Input type] Parameter input
[Data type] Bit

#0 PCE Program editing is:
0: Performed in the word edit mode.
1: Performed in the character edit mode.

#1 PDM On the Data Server file list screen:
0: M198 operation folders and DNC operation files can be set.
1: Folders in the Data Server can be set as the foreground folder and background folder.

NOTE
When an M198 external subprogram call or DNC operation is performed on the Data Server, set this bit to 0.
For the foreground and background folders, refer to Chapter, "PROGRAM MANAGEMENT".

	#7	#6	#5	#4	#3	#2	#1	#0
3280								NLC

[Input type] Parameter input
[Data type] Bit

#0 NLC Dynamic display language switching is:
0: Enabled.
1: Disabled.
When dynamic display language switching is disabled, the language setting screen is not displayed. In this case, change the setting of parameter No. 3281 on the parameter screen then turn on the power again to switch the display language.

3281	Display language

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 19
Select a display language from the following:
0 : English
1 : Japanese
2 : German
3 : French
4 : Chinese(traditional characters)
5 : Italian
6 : Korean
7 : Spanish
8 : Dutch
9 : Danish
10 : Portuguese
11 : Polish
12 : Hungarian
13 : Swedish
14 : Czech
15 : Chinese(simplified characters)
16 : Russian
17 : Turkish
18 : Bulgarian
19 : Rumanian
If a number not indicated above is set, English is selected.

	#7	#6	#5	#4	#3	#2	#1	#0
3400			PGD				MGC	

[Input type] Parameter input
[Data type] Bit path

#1 MGC When a single block specifies multiple M commands, an M code group check is:
0: Made.
1: Not made.

#5 PGD The G10.9 command (programmable diameter/radius specification switching) is:
0: Disabled.
1: Enabled.

NOTE
1 The option for the diameter and radius switching function is required.
2 When the G10.9 command is enabled by this parameter, signal-based diameter/radius switching is disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
3401	GSC	GSB	ABS	MAB				DPI
			ABS	MAB				DPI

[Input type] Parameter input
[Data type] Bit path

#0 DPI When a decimal point is omitted in an address that can include a decimal point
0: The least input increment is assumed. (Normal decimal point input)
1: The unit of mm, inches, degree, or second is assumed. (Pocket calculator type decimal point input)

#4 MAB Switching between the absolute and incremental programming in MDI operation
0: Performed by G90 or G91
1: Depending on the setting of bit 5 (ABS) of parameter No. 3401

NOTE
When G code system A of the lathe system is used, this parameter is invalid.

#5 ABS Program command in MDI operation
0: Assumed as an incremental programming
1: Assumed as an absolute programming

NOTE
ABS is valid when bit 4 (MAB) of parameter No. 3401 is set to 1.
When G code system A of the lathe system is used, this parameter is invalid.

#6 GSB The G code system is set.
#7 GSC

GSC	GSB	G code
0	0	G code system A
0	1	G code system B
1	0	G code system C

NOTE
G code system B and G code system C are optional functions. When no option is selected, G code system A is used, regardless of the setting of these parameters.

	#7	#6	#5	#4	#3	#2	#1	#0
3402	G23	CLR		FPM	G91			G01
	G23	CLR			G91	G19	G18	G01

[Input type] Parameter input
[Data type] Bit path

#0 G01 G01 Mode entered when the power is turned on or when the control is cleared
0: G00 mode (positioning)
1: G01 mode (linear interpolation)

#1 G18 Plane selected when power is turned on or when the control is cleared
0: G17 mode (plane XY)
1: G18 mode (plane ZX)

#2 G19 Plane selected when power is turned on or when the control is cleared
0: The setting of bit 1 (G18) of parameter No. 3402 is followed.
1: G19 mode (plane YZ)
When this bit is set to 1, set bit 1 (G18) of parameter No. 3402 to 0.

#3 G91 When the power is turned on or when the control is cleared
0: G90 mode (absolute programming)
1: G91 mode (incremental programming)

#4 FPM At power-on time or in the cleared state:
0: G99 or G95 mode (feed per revolution) is set.
1: G98 or G94 mode (feed per minute) is set.

#6 CLR RESET key on the MDI unit, external reset signal, reset and rewind signal, and emergency stop signal
0: Cause reset state.
1: Cause clear state.
For the reset and clear states, refer to Appendix in the OPERATOR'S MANUAL.

#7 G23 When the power is turned on
0: G22 mode (stored stroke check on)
1: G23 mode (stored stroke check off)

	#7	#6	#5	#4	#3	#2	#1	#0
3403		ADB						

[Input type] Parameter input
[Data type] Bit path

#6 ADB When the same address two or more times are specified in one block:
0: The address specified last is valid.
1: It is treated as a program error and the alarm PS5074, "ADDRESS DUPLICATION ERROR" is issued.

NOTE

The following notes apply when this parameter is set to 1:

1 When two or more M codes are acceptable to one block, up to three M codes can be specified in the same block. Specifying more than three results in the alarm PS5074.
2 You can specify any number of G codes in the same block as long as they belong to different groups. Specifying G codes belonging to the same group causes the alarm PS5074. You can however specify any number of G90 and G91 codes in the same block as they cause no alarm.
3 The alarm is not caused by blocks which call a custom macro or execution macro.
4 When G code system A is used with the lathe system, specifying an absolute programming and incremental programming for the same axis causes the alarm PS5074.

	#7	#6	#5	#4	#3	#2	#1	#0
3404	M3B		M02	M30		SBP	POL	

[Input type] Parameter input
[Data type] Bit path

#1 POL When a command is specified with a decimal point omitted in an address that can include a decimal point:

0: The command is assumed to be valid as it is.
1: A program error is assumed and the alarm PS5073, "NO DECIMAL POINT" is issued.

NOTE

The following notes apply when this parameter is set to 1:

1 G codes with a decimal point omitted do not cause the alarm PS5073.
2 Commands using a macro variable or numerical expression are treated as commands with a decimal point. Accordingly, they do not cause the alarm PS5073.
3 Argument specification I/II of a custom macro/execution macro does not cause the alarm PS5073.
4 Omitting a decimal point from a command of an extended axis name causes the alarm PS5073.
5 Omitting a decimal point from a command in an execution macro also causes the alarm PS5073.
6 Address R indicating setting data for programmable parameter input (G10L52) does not cause the alarm PS5073.

#2 SBP In an external device subprogram call, the address P format is based on:
0: File number specification
1: Program number specification

> **NOTE**
> In memory card operation, the program number specification format is used, regardless of the setting of this parameter.

#4 M30 When M30 is specified in a memory operation:
0: M30 is sent to the machine, and the head of the program is automatically searched for. So, when the ready signal FIN is returned and a reset or reset and rewind operation is not performed, the program is executed, starting from the beginning.
1: M30 is sent to the machine, but the head of the program is not searched for. (The head of the program is searched for by the reset and rewind signal.)

#5 M02 When M02 is specified in memory operation
0: M02 is sent to the machine, and the head of the program is automatically searched for. So, when the end signal FIN is returned and a reset or reset and rewind operation is not performed, the program is executed, starting from the beginning.
1: M02 is sent to the machine, but the head of the program is not searched for. (The head of the program is searched for by the reset and rewind signal.)

#7 M3B The number of M codes that can be specified in one block
0: One
1: Up to three

	#7	#6	#5	#4	#3	#2	#1	#0
3405				CCR	G36		DWL	AUX
							DWL	AUX

[Input type] Parameter input
[Data type] Bit path

#0 AUX When the second auxiliary function is specified in the calculator-type decimal point input format or with a decimal point, the multiplication factor for a value output (onto the code signal) relative to a specified value is such that:
0: The same multiplication factor is used for both of metric input and inch input.
1: A multiplication factor used for inch input is 10 times greater than that used for metric input.

When the second auxiliary function is specified in the calculator-type decimal point input format or with a decimal point, the value output onto the code signal is a specified value multiplied by a value indicated below.

Increment system		Parameter AUX=0	Parameter AUX=1
Metric input system	IS-A for reference axis	100 times	100 times
	IS-B for reference axis	1000 times	1000 times
	IS-C for reference axis	10000 times	10000 times
	IS-D for reference axis	100000 times	100000 times
	IS-E for reference axis	1000000 times	1000000 times
Inch input system	IS-A for reference axis	100 times	1000 times
	IS-B for reference axis	1000 times	10000 times
	IS-C for reference axis	10000 times	100000 times
	IS-D for reference axis	100000 times	1000000 times
	IS-E for reference axis	1000000 times	10000000 times

#1 DWL The dwell time (G04) is:
0: Always dwell per second.
1: Dwell per second in the feed per minute mode (G94), or dwell per rotation in the feed per rotation mode (G95).

#3 G36 As a G code to be used with the automatic tool length measurement function (M series)/automatic tool offset function (T series) is:
0: G36 (T series only)/G37 is used.
1: G37.1/G37.2/G37.3 is used.

NOTE
If it is necessary to perform circular threading (counterclockwise), set this parameter to 1.

#4 CCR Addresses used for chamfering
0: Address is “I”, “J”, or “K”.
In direct drawing dimension programming, addresses ",C", ",R", and ",A" (with comma) are used in stead of "C", "R", and "A".
1: Address is “C”.
Addresses used for direct drawing dimension programming are "C", "R", and "A" without comma.

NOTE
If this bit (CCR) is set to 0, the function for changing the compensation direction by specifying I, J, or K in a G01 block in the cutter compensation/ tool nose radius compensation mode cannot be used.
If this bit (CCR) is set to 1 when address C is used as an axis name, the chamfer function cannot be used.

	#7	#6	#5	#4	#3	#2	#1	#0
3406	C07	C06	C05	C04	C03	C02	C01	
	#7	#6	#5	#4	#3	#2	#1	#0
3407	C15	C14	C13	C12	C11	C10	C09	C08
	#7	#6	#5	#4	#3	#2	#1	#0
3408	C23	C22		C20	C19	C18	C17	C16
	#7	#6	#5	#4	#3	#2	#1	#0
3409		C30	C29	C28	C27	C26	C25	C24

[Input type] Parameter input
[Data type] Bit

C01 to C30 If bit 6 (CLR) of parameter No. 3402 is set to 1, set a group of G codes to be placed in the cleared state when the CNC is reset by the RESET key of the MDI unit, the external reset signal, the reset and rewind signal, or the emergency stop signal.
The table below indicates the correspondence between bits and G code groups
The setting of a bit has the following meaning:
0: Places the G code group in the cleared state.
1: Does not place G code group in the cleared state.

Parameter	G code group
C01	01
C02	02

Parameter	G code group
C03	03
:	:
C30	30

3410	Tolerance of arc radius

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
When a circular interpolation command is executed, the tolerance for the radius between the start point and the end point is set.

3411	M code preventing buffering 1
3412	M code preventing buffering 2
to	to
3420	M code preventing buffering 10

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999
Set M codes that prevent buffering the following blocks. If processing directed by an M code must be performed by the machine without buffering the following block, specify the M code.
M00, M01, M02, and M30 always prevent buffering even when they are not specified in these parameters.

3421	Range specification 1 of M codes that do not perform buffering (lower limit)
3422	Range specification 1 of M codes that do not perform buffering (upper limit)
3423	Range specification 2 of M codes that do not perform buffering (lower limit)
3424	Range specification 2 of M codes that do not perform buffering (upper limit)
3425	Range specification 3 of M codes that do not perform buffering (lower limit)
3426	Range specification 3 of M codes that do not perform buffering (upper limit)
3427	Range specification 4 of M codes that do not perform buffering (lower limit)
3428	Range specification 4 of M codes that do not perform buffering (upper limit)
3429	Range specification 5 of M codes that do not perform buffering (lower limit)
3430	Range specification 5 of M codes that do not perform buffering (upper limit)
3431	Range specification 6 of M codes that do not perform buffering (lower limit)
3432	Range specification 6 of M codes that do not perform buffering (upper limit)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999

When a specified M code is within the range specified with parameters Nos. 3421 and 3422, 3423 and 3424, 3425 and 3426, 3427 and 3428, 3429 and 3430, or 3431 and 3432, buffering for the next block is not performed until the execution of the block is completed.

NOTE
M00, M01, M02, and M30 are M codes that do not perform buffering, regardless of parameter setting.
M98, M99, M codes for calling subprograms, and M codes for calling custom macros are M codes that performs buffering, regardless of parameter setting.

3441	Start number of M codes for which an M code group can be set (1)
3442	Start number of M codes for which an M code group can be set (2)
3443	Start number of M codes for which an M code group can be set (3)
3444	Start number of M codes for which an M code group can be set (4)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0, 100to99999999

Code numbers 0 to 99 on the M code group setting screen correspond to M00 to M99. When adding M codes after the first 100 M codes, specify a start M code number in these parameters. Thus, up to 400 M codes can be added to the M code group setting screen in groups of 100 M codes starting with the set value. When 0 is set, no M codes are added to the M code group setting screen.
When setting these parameters, follow the setting condition described below. If the condition is not satisfied, no M codes are added to the M code group setting screen as in the case where 0 is set.
(Setting condition)
The settings of parameters (1) to (4) (excluding the setting of 0) must satisfy:
99 < (1), (1)+99 < (2), (2)+99 < (3), (3) +99 < (4)

	#7	#6	#5	#4	#3	#2	#1	#0
3450	BDX							AUP

[Input type] Parameter input
[Data type] Bit path

#0 AUP The second auxiliary function specified in the calculator-type decimal point input format, with a decimal point, or with a negative value is:
0: Disabled.
1: Enabled.
If the second auxiliary function is specified after setting this bit to 0, the following operation results:
1. When a value is specified without a decimal point
   A specified value is output onto the code signal without modification, regardless of the setting of the calculator-type decimal point input format (with bit 0 (DPI) of parameter No. 3401).
2. When a value is specified with a decimal point
   The alarm PS0007, “ILLEGAL USE OF DECIMAL POINT” is issued.
3. When a negative value is specified
   The alarm PS0006, “ILLEGAL USE OF MINUS SIGN” is issued.

#7 BDX When ASCII code is called using the same address as the address for the second auxiliary function (specified by parameter No. 3460), this parameter prevents the argument unit used when the option for the second auxiliary function is selected from differing from the argument unit used when the same option is not selected.

0: When bit 0 (AUP) of parameter No. 3450 is set to 1, the argument unit differs, depending on whether the option for the second auxiliary function is selected or not.

1: The same argument unit is used. (The unit applied when the option for the second auxiliary function is selected is used.)

[Example]

A setting is made so that address B is used to call O9004, and the program O1 below is executed with parameter No. 3460 = 66.

```
O1          O9004
B2          #500 = #146
M30         M99
```

When the increment system is IS-B, and metric input is used, #500 assumes a value indicated in the table below.

Bit 0 (DPI) of parameter No. 3401	Bit 0 (AUP) of parameter No. 3450	BDX=0		BDX=1
		Without the second auxiliary function option	With the second auxiliary function option	
0	0	2.000	2.000	2.000
	1	2.000	0.002	0.002
1	0	2.000	2.000	2.000
	1	2.000	2.000	2.000

	#7	#6	#5	#4	#3	#2	#1	#0
3451								
								GQS

[Input type] Parameter input
[Data type] Bit path

#0 GQS When threading is specified, the threading start angle shift function (Q) is:

0: Disabled.

1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
3452	EAP							
	EAP							

[Input type] Parameter input
[Data type] Bit path

#7 EAP When bit 0 (ADX) of parameter No. 3455 is set to 1, calculator-type decimal point input at a macro calling argument address is:

0: Enabled.

1: Disabled.

NOTE
This parameter is valid when bit 0 (DPI) of parameter No. 3401 is set to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
3454				G1B		DTO		

[Input type] Parameter input
[Data type] Bit path

#2 DTO The method of specifying a rotation axis in cylindrical interpolation mode is set.
0: In cylindrical interpolation mode, the rotation axis is specified by angle.
1: In cylindrical interpolation mode, the rotation axis is specified by distance on an expanded plane.

#4 G1B In programmable parameter input, specifying a change to a specific bit parameter is:
0: Disabled.
1: Enabled. (A bit number is specified with Q_.)

	#7	#6	#5	#4	#3	#2	#1	#0
3455								AXDx

[Input type] Parameter input
[Data type] Bit axis

#0 AXDx If a decimal point is omitted for an axis address with which a decimal point can be used, the value is determined:
0: In accordance with the least input increment. (Normal decimal point input)
1: In millimeters, inches, degrees, or seconds. (calculator-type decimal point input)

NOTE
This parameter specifies the calculator-type decimal point input function for each axis.
For the same axis name, be sure to make the same setting.

	#7	#6	#5	#4	#3	#2	#1	#0
3456								PVT

[Input type] Parameter input
[Data type] Bit axis

#0 PVT As a pivot axis control axis:
0: Not used.
1: Used.

NOTE
1 When this parameter is set, the power must be turned off before operation is continued.
2 The option for pivot axis control is required.
3 When an axis uses a synchronized axis, make this setting for both of them.

	#7	#6	#5	#4	#3	#2	#1	#0
3457	SCF	SCC			SYS	MC1	MC2	LIB

[Input type] Parameter input
[Data type] Bit path

NOTE
1 The parameters LIB, MC2, MC1, and SYS are used to set a search folder for the following subprogram/macro calls:
- Subprogram call based on an M code
- Subprogram call based on a particular address
- Subprogram call based on a second auxiliary function code
- Macro call based on a G code
- Macro call based on an M code
- Macro call based on a T code
- One-touch macro call

2 The parameter SCF is used to set whether to add a search folder for the following subprogram/macro calls:
- Subprogram call based on M98
- Figure copy based on G72.1/G72.2
- Macro call based on G65/G66/G66.1
- Macro interrupt based on M96

#0 LIB The common program directory "//CNC_MEM/USER/LIBRARY/" of the initial directories is:
0: Set as a search directory.
1: Not set as a search directory.

#1 MC2 MTB dedicated directory 2 "//CNC_MEM/MTB2/" of the initial directories is:
0: Set as a search directory.
1: Not set as a search directory.

#2 MC1 MTB dedicated directory 1 "//CNC_MEM/MTB1/" of the initial directories is:
0: Set as a search directory.
1: Not set as a search directory.

#3 SYS The system directory "//CNC_MEM/SYSTEM/" of the initial directories is:
0: Set as a search directory.
1: Not set as a search directory.

#6 SCC The same folder as the main program is added to the top of the search order as a search folder for the following each subprogram call and macro call.
- Subprogram call by M code
- Subprogram call by ASCII code
- Subprogram call by the second auxiliary function code
- Macro call by S code
- Macro call by T code
- Macro call by G code
- Macro call by M code
- One-touch macro call

The same folder as the main program is:
0: Not added in the search order.
1: Added in the search order.
When a search folder is added, a search is made in the following order:
0) Folder only for embedded macro (With the embedded macro-function.)
1) Folder where the main program is stored
2) Common program folder, which is an initial folder (LIBRARY)

3) MTB-dedicated folder 2, which is an initial folder (MTB2)
4) MTB-dedicated folder 1, which is an initial folder (MTB1)
5) System folder, which is an initial folder (SYSTEM)

The folders of 2) through 5) can be excluded from search target folders by setting the bits 0 (LIB), 1 (MC2), 2 (MC1), and 3 (SYS) of parameter No. 3457.

#7 SCF A search folder is:
0: Not added.
1: Added.

When a search folder is added, a search is made in the following order:
0) Folder only for embedded macro (With the embedded macro-function.)
1) Folder where the main program is stored
2) Common program folder, which is an initial folder (LIBRARY)
3) MTB-dedicated folder 2, which is an initial folder (MTB2)
4) MTB-dedicated folder 1, which is an initial folder (MTB1)
5) System folder, which is an initial folder (SYSTEM)

The folders of 3) through 5) can be excluded from search target folders by setting the bits 1 (MC2), 2 (MC1), and 3 (SYS) of parameter No. 3457.

	#7	#6	#5	#4	#3	#2	#1	#0
3458								TPS

[Input type] Parameter input
[Data type] Bit path

#0 TPS When a plane is selected on the lathe system in the power-on state or cleared state:
0: G18 mode (Z-X plane) is selected.
1: Bits 1 (G18) and 2 (G19) of parameter No. 3402 are followed.

	#7	#6	#5	#4	#3	#2	#1	#0
3459								ESL

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 ESL When an NC program contains lowercase alphabetic characters:
0: An alarm is issued.
1: The lowercase characters are converted into their uppercase equivalents.

This parameter specifies how to handle lowercase alphabetic characters included in an NC program, except in its comment part, program name, and folder name, when the program is registered from an external device into CNC built-in memory or compared. The parameter setting also applies to M198 operation or DNC operation as automatic operation.

[Example] G90G01X100y50;
When ESL is 1, the program is assumed to be G90G01X100Y50;.

When ESL is 0, the alarm SR1090, "PROGRAM FORMAT ERROR" is displayed upon registration or comparison. During operation, the alarm PS1090, "PROGRAM FORMAT ERROR" is issued.

NOTE
1 Program transfer by the program batch input/output function is excluded.
2 Program transfer by the FTP file transfer function is excluded.

3460 **Second auxiliary function specification address**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 65to67, 85to87

Specify which of A, B, C, U, V, and W is to be used as the address for specifying the second auxiliary function. If an address used as an axis name is specified, the second auxiliary function is disabled.

Name	A	B	C	U	V	W
Setting value	65	66	67	85	86	87

Address B is assumed when a value other than the above is set.
However, the name U, V, or W can be used with the T series only when G code system B or C is used. When a value from 85 to 87 is specified with G code system A, the specification address for the second auxiliary function is B.

3471 **Allowable difference between the specified end position and the end position obtained from the increase/decrease and frequency in spiral interpolation or conic interpolation**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the maximum allowable difference (absolute value) between the specified end position and the end position obtained from the increase/decrease and frequency in spiral or conic interpolation.

3472 **Minimum radius needed to maintain the actual speed in spiral or conic interpolation**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] (For IS-B and millimeter machines, 1.0 to 999999.999; for inch machines, 1.0 to 99999.9999)

If this parameter value is 0 or a value outside the valid data range, the minimum value of the range is assumed.

In spiral interpolation and conic interpolation, the speed is generally held constant. In an area near the center, the spiral radius decreases, resulting in an extremely high angular velocity. To prevent this, once the spiral radius has reached the parameter-set value, the angular velocity subsequently remains constant. As a result, the actual speed decreases.

	#7	#6	#5	#4	#3	#2	#1	#0
3602								APE

[Input type] Parameter input
[Data type] Bit

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 APE The input type of Stored Pitch Error Compensation data is
0: An incremental value.
1: A total value.

This function is effective to the following functions.
- Stored Pitch Error Compensation
- Bi-directional Pitch Error Compensation
- Interpolation Type Pitch Error Compensation
- Periodical Secondary Pitch Error Compensation
- Interpolation Type Straightness Compensation
- Spindle Command Synchronous Control Independent Pitch Error Compensation

NOTE
If this parameter is changed, the data of stored pitch error compensation is cleared automatically at next power on.

	#7	#6	#5	#4	#3	#2	#1	#0
3605								BDPx

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 BDPx Both-direction pitch error compensation is:
0: Not used.
1: Used.

3620	Number of the pitch error compensation position for the reference position for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1535
Set the number of the pitch error compensation position for the reference position for each axis.

3621	Number of the pitch error compensation position at extremely negative position for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1535
Set the number of the pitch error compensation position at the extremely negative position for each axis.

3622	Number of the pitch error compensation position at extremely positive position for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1535
Set the number of the pitch error compensation position at the extremely positive position for each axis.
This value must be larger than set value of parameter No. 3620.

3623	Magnification for pitch error compensation for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 100
Set the magnification for pitch error compensation for each axis.
If the magnification is set to 1, the same unit as the detection unit is used for the compensation data.
If 0 is set, compensation is not performed.

3624	Interval between pitch error compensation positions for each axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] See the description below.
The pitch error compensation positions are arranged with equal spacing. The space between two adjacent positions is set for each axis. The minimum interval between pitch error compensation positions is limited and obtained from the following equation:

Minimum interval between pitch error compensation positions = maximum feedrate/7500
Unit : mm, inch, deg or mm/min, inch/min, deg/min

[Example] When the maximum feedrate is 15000 mm/min, the minimum interval between pitch error compensation positions is 2 mm.

3625	Travel distance per revolution in pitch error compensation of rotation axis type

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] See the description below.

If the pitch error compensation of rotation axis type is performed (bit 1 (ROSx) of parameter No. 1006 is set to 0 and bit 0 (ROTx) of parameter No. 1006 is set to 1), set the travel distance per revolution. The travel distance per revolution does not have to be 360 degrees, and a cycle of pitch error compensation of rotation axis type can be set.
However, the travel distance per revolution, compensation interval, and number of compensation points must satisfy the following condition:
(Travel distance per revolution)
= (Compensation interval) × (Number of compensation points)
The compensation at each compensation point must be set so that the total compensation per revolution equals 0.

NOTE
If 0 is set, the travel distance per revolution becomes 360 degrees.

3626	Number of the both-direction pitch error compensation position at extremely negative position (for movement in the negative direction)

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 1023, 3000 to 4023

When using both-direction pitch error compensation, set the number of compensation point at the farthest end in the negative direction for a movement in the negative direction.

NOTE
1 For a movement in the positive direction, set the compensation point number at the farthest end in the negative direction in parameter No. 3621.
2 A set of compensation data items for a single axis should not be set to lie astride 1023 to 3000.

3627	Pitch error compensation at reference position when a movement to the reference position is made from the direction opposite to the direction of reference position return

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] -32768 to 32767

Set the absolute value of pitch error compensation at reference position when a movement to the reference position is made from the negative direction if the direction of reference position return (bit 5 (ZMI) of parameter No. 1006) is positive or from the positive direction if the direction of reference position return is negative.

	#7	#6	#5	#4	#3	#2	#1	#0
3700							NRF	CRF

[Input type] Parameter input
[Data type] Bit path

#0 CRF Reference position setting at an arbitrary position under Cs contour control is:
0: Not used.
1: Used.

NOTE
When this function is used, an attempt to specify G00 for a Cs contour control axis without performing a reference position return operation even once after switching the serial spindle to the Cs contour control mode results in the alarm PS0303, “REFERENCE POSITION RETURN IS NOT PERFORMED” even if bit 1 (NRF) of parameter No. 3700 is set to 0. Be sure to perform a reference position return operation by specifying G28.

#1 NRF With the first move command (G00) after switching the series spindle to Cs contour control mode:
0: A reference position return operation is once performed then positioning is performed.
1: A normal positioning operation is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
3702							EMS	

[Input type] Parameter input
[Data type] Bit path

#1 EMS The multi-spindle control function is:
0: Used.
1: Not used.

	#7	#6	#5	#4	#3	#2	#1	#0
3716								A/Ss

[Input type] Parameter input
[Data type] Bit spindle

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 A/Ss Spindle motor type is :
0: Analog spindle.
1: Serial spindle.

NOTE
1 When an analog spindle is used, the option for spindle analog output is required.
2 When a serial spindle is used, the option for spindle serial output is required.
3 The option for the number of controlled spindles needs to be specified.

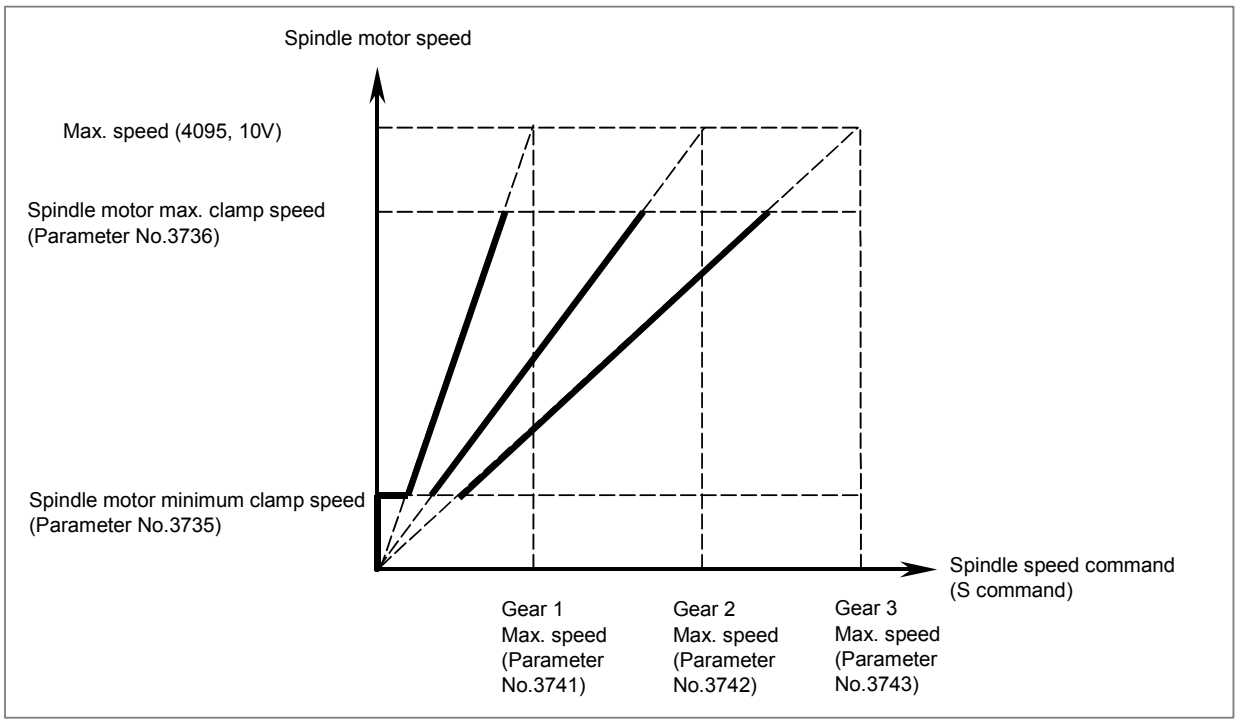
3717	Motor number to each spindle

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to Maximum number of controlled axes
Set a spindle amplifier number to be assigned to each spindle.
0: No spindle amplifier is connected.
1: Spindle motor connected to amplifier number 1 is used.
2: Spindle motor connected to amplifier number 2 is used.
to
n: Spindle motor connected to amplifier number n is used.

3741	Maximum spindle speed for gear 1
3742	Maximum spindle speed for gear 2
3743	Maximum spindle speed for gear 3
3744	Maximum spindle speed for gear 4

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999
Set the maximum spindle speed corresponding to each gear.



3770	Axis as the calculation reference in constant surface speed control

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the axis as the calculation reference in constant surface speed control.

NOTE
When 0 is set, constant surface speed control is always applied to the X-axis. In this case, specifying P in a G96 block has no effect on the constant surface speed control.

3781	P code for selecting the spindle in multi-spindle control

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 32767
If bit 3 (MPP) of parameter No. 3703 is set to 1, set the P code to select each spindle under multi-spindle control. Specify the P code in a block containing the S command.

[Example] If the P code value for selecting the second spindle is set to 3,
S1000 P3;
causes the second spindle to rotate at S1000.

Parameters Nos. 4000 to 4799 are used in the processing on the serial spindle amplifier. For details of these parameters, refer to either of the following manuals, depending on the serial spindle that is actually used.

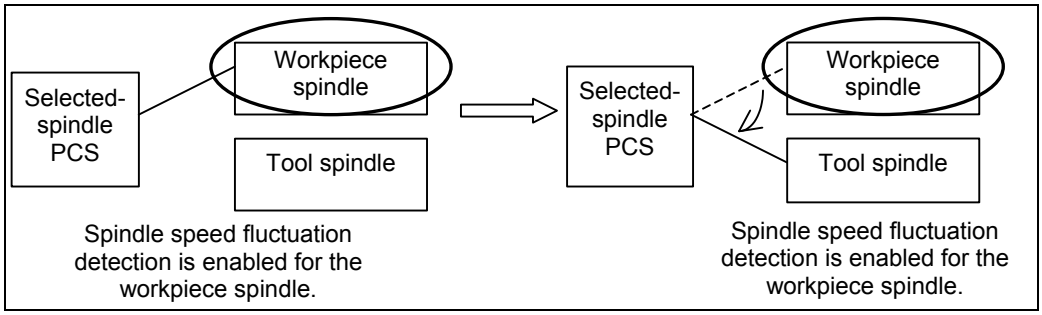
- FANUC AC SPINDLE MOTOR αi series Parameter Manual (B-65280EN)

	#7	#6	#5	#4	#3	#2	#1	#0
4900	FDTs			FDEs				FLRs

[Input type] Parameter input
[Data type] Bit spindle

#0 FLRs When the spindle speed fluctuation detection function is used, the unit of an allowable ratio (q) and fluctuation ratio (r) set by parameters No. 4911 and No. 4912 is:
0: 1%
1: 0.1%

#4 FDEs Spindle speed fluctuation detection function is:
0: Enabled.
1: Disabled.
If the position coder selection signal is selected for a spindle for which this parameter is 1, the target spindle of spindle speed fluctuation detection remains unchanged. Spindle speed fluctuation detection stays enabled for the spindle for which spindle speed fluctuation detection was enabled before the selection of the position coder selection signal.
An example is given below.
[Example]
Bit 4 (FDE) of parameter No. 4900 (Workpiece spindle) = 0
Bit 4 (FDE) of parameter No. 4900 (Tool spindle) = 1
Even when the position coder selection signal is switched from the workpiece spindle to the tool spindle, the target spindle of spindle speed fluctuation detection remains unchanged, that is, the function stays enabled for the workpiece spindle.



Note) The selected spindle varies depending on the state of the position coder selection signal.

NOTE
1 If bit 4 (FDE) of parameter No. 4900 is 0 for all spindles, spindle speed fluctuation detection is enabled for the spindle selected with the position coder selection signal as is conventionally. If the parameter FDE is 1 for all spindles, spindle speed fluctuation detection is enabled for the spindle selected with the position coder selection signal.
2 When the parameter FDE is 0 for all spindles, setting the parameter FDE for the spindle selected with the position coder selection signal to 1 does not cause an immediate change to the target spindle, which stays as the target until the next position coder selection signal is issued.
3 If the parameter FDE for the first spindle is 1 and the first spindle is the tool spindle when the power is turned on, spindle speed fluctuation detection remains enabled for the first spindle until a spindle for which the parameter FDE is 0 is selected.

#7 FDTs Spindle speed fluctuation detection is started:
0: When the actual spindle speed reaches a specified range or when the time specified with parameter No. 4914 elapses.
1: After the time specified with parameter No. 4914 has elapsed.

4911 **Allowable speed ratio (q) used to assume that the spindle has reached a specified speed**

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%, 0.1%
[Valid data range] 1 to 100, 1 to 1000
When the spindle speed fluctuation detection function is used, set an allowable speed ratio (q) used to assume that the spindle has reached a specified speed.

NOTE
The unit of data is determined by bit 0 (FLR) of parameter No. 4900.

4912 **Spindle fluctuation ratio (r) for not issuing a spindle speed fluctuation detection alarm**

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%, 0.1%
[Valid data range] 1 to 100, 1 to 1000
When the spindle speed fluctuation detection function is used, set a spindle fluctuation ratio (r) for not issuing an alarm.

NOTE
The unit of data is determined by bit 0 (FLR) of parameter No. 4900.

4913 **Spindle speed fluctuation width (i) for not issuing a spindle speed fluctuation detection alarm**

[Input type] Parameter input
[Data type] 2-word spindle

[Unit of data] min^{-1}
[Valid data range] 0 to 99999
When the spindle speed fluctuation detection function is used, set an allowable fluctuation width (i) for not issuing an alarm.

4914	Time (p) from the change of a specified speed until spindle speed fluctuation detection is started

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] msec
[Valid data range] 0 to 999999
When the spindle speed fluctuation detection function is used, set a time (p) from the change of a specified speed until spindle speed fluctuation detection is started. In other words, spindle speed fluctuation detection is not performed until a set time has elapsed after a specified speed is changed. However, when the actual spindle speed is assumed to have reached a specified value within a set time (p), spindle speed fluctuation detection is started.

	#7	#6	#5	#4	#3	#2	#1	#0
4950						ISZs	IDMs	IORs

[Input type] Parameter input
[Data type] Bit spindle

#0 IORs Resetting the system in the spindle positioning mode
0: Does not release the mode.
1: Releases the mode

#1 IDMs The direction of spindle positioning (half-fixed angle positioning based on M codes) is:
0: Plus direction.
1: Minus direction.

#2 ISZs When an M code for switching to the spindle positioning mode is specified for spindle positioning:
0: The spindle is switched to the spindle positioning mode, and spindle orientation operation is performed.
1: Only the switching of the spindle to the spindle positioning mode is performed. (Spindle orientation operation is not performed.)

4960	M code specifying the spindle orientation

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 6 to 97
Set an M code for switching to the spindle positioning mode.

NOTE
1 Do not set an M code that duplicates other M codes used for spindle positioning.
2 Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

4961	M code releasing the spindle positioning mode

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 6 to 97
Set an M code for canceling the spindle positioning mode on the spindle positioning axis.

NOTE
1 Do not set an M code that duplicates other M codes used for spindle positioning.
2 Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

4962	M code for specifying a spindle positioning angle

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 6 to 9999999
Two methods are available for specifying spindle positioning. One method uses axis address for arbitrary-angle positioning. The other use an M code for half-fixed angle positioning. This parameter sets an M code for the latter method.
In this parameter, set an M code to be used for half-fixed angle positioning based on M codes.
Six M code from Mα to M(α+5) are used for half-fixed angle positioning, when α is the value of this parameter.

- When the number of M codes is set in parameter No. 4964, let α be the value set in parameter No. 4962, and let β be the value set in parameter No. 4964. Then, β M codes from Mα to M(α+β-1) are used as M codes for half-fixed angle positioning based on M codes.

The table below indicates the relationship between the M codes and positioning angles.

M code	Positioning angle	Example: Positioning angle when θ = 30°
Mα	θ	30°
M(α+1)	2θ	60°
M(α+2)	3θ	90°
M(α+3)	4θ	120°
M(α+4)	5θ	150°
M(α+5)	6θ	180°
:	:	:
M(α+β-1)	β×θ	β×30°

β represents the number of M codes set in parameter No. 4964.
(When parameter No. 4964 is set to 0, β = 6.)
θ represents the basic angular displacement set in parameter No. 4963.

NOTE
1 Do not set an M code that duplicates other M codes used for spindle positioning.
2 Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

4963	Basic angle for half-fixed angle positioning

[Input type] Parameter input

[Data type] Real spindle
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 to 60

This parameter sets a basic angular displacement used for half-fixed angle positioning using M codes.

4964	Number of M codes for specifying a spindle positioning angle

[Input type] Parameter input
[Data type] 2-word spindle
[Valid data range] 0 to 255

This parameter sets the number of M codes used for Half-fixed angle positioning using M codes.

As many M codes as the number specified in this parameter, starting with the M code specified in parameter No. 4962, are used to specify half-fixed angle positioning.

Let α be the value of parameter No. 4962, and let β be the value of parameter No. 4964. That is, M codes from Mα to M(α+β-1) are used for half-fixed angle positioning.

Setting this parameter to 0 has the same effect as setting 6. That is, M code from Mα to M(α+5) are used for half-fixed angle positioning.

NOTE
1 Make sure that M codes from Mα to M (α+β-1) do not duplicate other M codes.
2 Do not set an M code that duplicates other M codes used for spindle positioning.
3 Do not set an M code used with other functions (such as M00-05, 30, 98, and 99, and M codes for calling subprograms).

	#7	#6	#5	#4	#3	#2	#1	#0
5001		EVO						
		EVO			TAL		TLB	TLC

[Input type] Parameter input
[Data type] Bit path

#0 TLC

#1 TLB These bits are used to select a tool length compensation type.

Type	TLB	TLC
Tool length compensation A	0	0
Tool length compensation B	1	0
Tool length compensation C	-	1

The axis to which cutter compensation is applied varies from type to type as described below.
Tool length compensation A : Z-axis at all times
Tool length compensation B : Axis perpendicular to a specified plane (G17/G18/G19)
Tool length compensation C : Axis specified in a block that specifies G43/G44

#3 TAL Tool length compensation C
0: Generates an alarm when two or more axes are offset
1: Not generate an alarm even if two or more axes are offset

#6 EVO If a tool compensation value modification is made for tool length compensation A or tool length compensation B in the offset mode (G43 or G44):
0: The new value becomes valid in a block where G43, G44, or an H code is specified next.
1: The new value becomes valid in a block where buffering is performed next.

	#7	#6	#5	#4	#3	#2	#1	#0
5002						LWT	LGN	

[Input type] Parameter input
[Data type] Bit path

#1 LGN Geometry offset number of tool offset
0: Is the same as wear offset number
1: Specifies the geometry offset number by the tool selection number

NOTE
This parameter is valid when the option for tool geometry/wear compensation is specified.

#2 LWT Tool wear compensation is performed by:
0: Moving the tool.
1: Shifting the coordinate system.

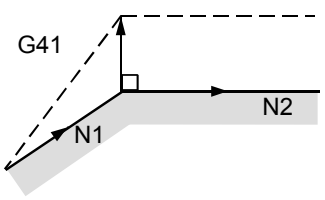
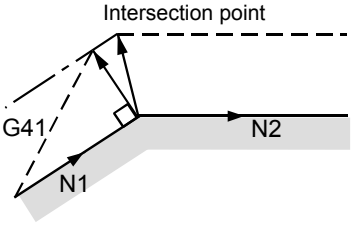
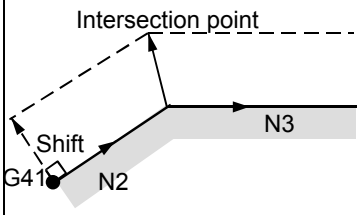
NOTE
This parameter is valid when the option for tool geometry/wear compensation is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
5003							SUV	SUP

[Input type] Parameter input
[Data type] Bit path

#0 SUP

#1 SUV These bits are used to specify the type of startup/cancellation of tool radius - tool nose radius compensation.

SUV	SUP	Type	Operation
0	0	Type A	A compensation vector perpendicular to the block next to the startup block or the block preceding the cancellation block is output. 
0	1	Type B	A compensation vector perpendicular to the startup block or cancellation block and an intersection vector are output. 
1	0 1	Type C	When the startup block or cancellation block specifies no movement operation, the tool is shifted by the cutter compensation amount in a direction perpendicular to the block next to the startup or the block before cancellation block.  When the block specifies movement operation, the type is set according to the SUP setting; if SUP is 0, type A is set, and if SUP is 1, type B is set.

NOTE
When SUV,SUP = 0,1 (type B), an operation equivalent to that of FS16*i*-T is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
5004							ORC	
						ODI		

[Input type] Parameter input
[Data type] Bit path

#1 ORC The setting of a tool offset value is corrected as:
0: Diameter value
1: Radius value

NOTE
This parameter is valid only for an axis based on diameter specification. For an axis based on radius specification, specify a radius value, regardless of the setting of this parameter.

#2 ODI The setting of a tool radius - tool nose radius compensation value is corrected as:
0: Radius value
1: Diameter value

	#7	#6	#5	#4	#3	#2	#1	#0
5008				MCR				

[Input type] Parameter input
[Data type] Bit path

#4 MCR If G41/G42 (tool radius - tool nose radius compensation) is specified in the MDI mode, an alarm is:
0: Not raised.
1: Raised. (alarm PS5257, “G41/G42 NOT ALLOWED IN MDI MODE”)

	Number of digits of an offset number used with a T code command
5028	

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3
Specify the number of digits of a T code portion that is used for a tool offset number (wear offset number when the tool geometry/wear compensation function is used).
When 0 is set, the number of digits is determined by the number of tool compensation values.
When the number of tool compensation values is 1 to 9: Lower 1 digit
When the number of tool compensation values is 10 to 99: Lower 2 digits
When the number of tool compensation values is 100 to 999: Lower 3 digits
[Example] When an offset number is specified using the lower 2 digits of a T code, set 2 in parameter No. 5028.
Txxxxxx yy
xxxxxx : Tool selection
yy : Tool offset number

NOTE
A value longer than the setting of parameter No. 3032 (allowable number of digits of a T code) cannot be set.

	Number of tool compensation value memories common to paths
5029	

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 999

When using memories common to paths, set the number of common tool compensation values in this parameter.
Ensure that the setting of this parameter does not exceed the number of tool compensation values set for each path (parameter No. 5024).

[Example 1] When parameter No. 5029 = 10, parameter No. 5024 (path 1) = 15, and parameter No. 5024 (path 2) = 30 in a 2-path system, tool compensation numbers 1 to 10 of all paths are made common.

[Example 2] When parameter No. 5029 = 20 and the other conditions are the same as for Example 1, tool compensation numbers 1 to 15 are made common.

NOTE
1 When a multi-path system involving the machining center system and lathe system is used, memories are made common in each system.
2 In each of the machining center system and lathe system, the same unit of tool compensation values needs to be used.
3 Ensure that the setting of parameter No. 5029 does not exceed the number of tool compensation values for each path (parameter No. 5024). If the setting of parameter No. 5029 exceeds the number of compensation values of a path, the least of the numbers of compensation values in all paths is made common.
4 When 0 or a negative value is set, memories common to paths are not used.

	#7	#6	#5	#4	#3	#2	#1	#0
5040					TCT			OWD

[Input type] Parameter input
[Data type] Bit path

#0 OWD In radius programming (bit 1 (ORC) of parameter No. 5004 is set to 1),
0: Tool offset values of both geometry compensation and wear compensation are specified by radius.
1: Tool offset value of geometry compensation is specified by radius and tool offset value of wear compensation is specified by diameter, for an axis of diameter programming.

NOTE
This parameter is valid when the option for tool geometry/wear compensation is specified.

#3 TCT The tool change method is based on:
0: Turret rotation. (Tool change operation is performed with a T command only.)
With a T command, an auxiliary function and tool offset operation are performed.
1: Automatic tool changer (ATC).
(Tool change operation is performed with an M command (such as M06)).
With a T command, an auxiliary function only is performed.
This parameter is valid with a lathe system only.

> **⚠ WARNING**
> Before changing the setting of this parameter, cancel the offset. If the setting is changed while the offset is applied, the subsequent offset operation may not be performed correctly or an alarm PS0368 occurs.

	#7	#6	#5	#4	#3	#2	#1	#0
5042					OFE	OFD	OFC	OFA

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 OFA
#1 OFC
#2 OFD
#3 OFE These bits are used to specify the increment system and valid data range of a tool offset value.

For metric input

OFE	OFD	OFC	OFA	Unit	Valid data range
0	0	0	1	0.01mm	±9999.99mm
0	0	0	0	0.001mm	±9999.999mm
0	0	1	0	0.0001mm	±9999.9999mm
0	1	0	0	0.00001mm	±9999.99999mm
1	0	0	0	0.000001mm	±999.999999mm

For inch input

OFE	OFD	OFC	OFA	Unit	Valid data range
0	0	0	1	0.001inch	±999.999inch
0	0	0	0	0.0001inch	±999.9999inch
0	0	1	0	0.00001inch	±999.99999inch
0	1	0	0	0.000001inch	±999.999999inch
1	0	0	0	0.0000001inch	±99.9999999inch

5043	Axis number for which Y-axis offset is used

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 24
Set the number of an axis for which the tool offset is corrected.
If 0 or a value beyond the valid data range is set, the Y-axis offset is applied to the Y-axis of the basic three axes. If setting is made for the X- or Z-axis of the basic three axes, the standard tool offset for the X- or Z-axis is not used, and only the Y-axis offset is used.

5071	
	Number of first axis for grinding –wheel wear compensation

5072	
	Number of second axis for grinding–wheel wear compensation

[Input type] Parameter Input
[Data type] Byte path
[Valid data range] 1 to number of controlled axis
This parameter specifies the controlled axis numbers of the first and second axis for which grinding-wheel wear compensation is applied.

5081	1st-axis coordinate value of compensation center 1 in grinding-wheel wear compensation
5082	2nd-axis coordinate value of compensation center 1 in grinding-wheel wear compensation
5083	1st-axis coordinate value of compensation center 2 in grinding-wheel wear compensation
5084	2nd-axis coordinate value of compensation center 2 in grinding-wheel wear compensation
5085	1st-axis coordinate value of compensation center 3 in grinding-wheel wear compensation
5086	2nd-axis coordinate value of compensation center 3 in grinding-wheel wear compensation

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the coordinate value (in the workpiece coordinate system) of the center of compensation in grinding-wheel wear compensation.

	#7	#6	#5	#4	#3	#2	#1	#0
5101								FXY

[Input type] Parameter input
[Data type] Bit path

#0 FXY The drilling axis in the drilling canned cycle, or cutting axis in the grinding canned cycle is:

0: In case of the Drilling canned cycle:
Z-axis at all times.
In case of the Grinding canned cycle:
- For the Lathe system
Z-axis at all times.
- For the Machining Center system
G75,G77 command :Y-axis
G78,G79 command :Z-axis

1: Axis selected by the program

NOTE
In the case of the T series, this parameter is valid only for the drilling canned cycle in the Series 15 format.

	#7	#6	#5	#4	#3	#2	#1	#0
5105			TFA					

#5 TFA During tool center point control or tool length compensation in tool axis direction, canned cycles:

0: Cannot be used.

1: Can be used. However, an alarm PS5424, “ILLEGAL TOOL DIRECTION” is issued if the position of the rotation axis is not ±90° × n (n=0, 1, 2, ...) in the workpiece coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
5106								GFX

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 GFX When the options of multiple respective canned cycle and grinding canned cycle are both specified, G71/G72/G73/G74 commands are:

0: Multiple respective canned cycle.

1: Grinding canned cycle.

	#7	#6	#5	#4	#3	#2	#1	#0
5107				ICS				

[Input type] Parameter input
[Data type] Bit path

#4 ICS In-position check switching for drilling canned cycles is:

0: Disabled.

1: Enabled.

5176	Grinding axis number in Traverse Grinding Cycle(G71)
	Grinding axis number in Plunge Grinding Cycle(G75)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

For the Lathe system:

Set the Grinding axis number of Traverse Grinding Cycle(G71).

For the Machining Center system:

Set the Grinding axis number of Plunge Grinding Cycle(G75).

NOTE
The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, an alarm PS0456, “ILLEGAL PARAMETER IN GRINDING” is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, the alarm PS0456 is also issued.

5177	Grinding axis number of Traverse direct constant-size Grinding cycle(G72)
	Grinding axis number of Direct Constant Dimension Plunge Grinding Cycle(G77)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
For the Lathe system:
Set the Grinding axis number of Traverse direct constant-size Grinding cycle(G72).
For the Machining Center system:
Set the Grinding axis number of Direct Constant Dimension Plunge Grinding Cycle (G77).

NOTE
The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, an alarm PS0456, “ILLEGAL PARAMETER IN GRINDING” is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, the alarm PS0456 is also issued.

5178	Grinding axis number of Oscillation Grinding Cycle(G73)
	Grinding axis number of Continuous feed surface grinding cycle(G78)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
For the Lathe system:
Set the Grinding axis number of Oscillation Grinding Cycle(G73).
For the Machining Center system:
Set the Grinding axis number of Continuous feed surface grinding cycle(G78).

NOTE
The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, an alarm PS0456, “ILLEGAL PARAMETER IN GRINDING” is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, the alarm PS0456 is also issued.

5179	Grinding axis number of Oscillation Direct Fixed Dimension Grinding Cycle(G74)
	Grinding axis number of Intermittent feed surface grinding cycle(G79)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
For the Lathe system:
Set the Grinding axis number of Oscillation Direct Fixed Dimension Grinding Cycle(G74).
For the Machining Center system:
Set the Grinding axis number of Intermittent feed surface grinding cycle(G79).

NOTE
The axis number except for the cutting axis can be specified. When the axis number which is same to cutting axis is specified, an alarm PS0456, “ILLEGAL PARAMETER IN GRINDING” is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0, the alarm PS0456 is also issued.

5180	
	Axis number of dressing axis in Plunge grinding cycle(G75)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the axis number of dressing axis in Plunge grinding cycle(G75).

NOTE
The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, an alarm PS0456, “ILLEGAL PARAMETER IN GRINDING” is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the alarm PS0456 is also issued.

5181	
	Axis number of dressing axis in Direct constant dimension plunge grinding cycle(G77)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the axis number of dressing axis in Direct constant dimension plunge grinding cycle(G77).

NOTE
The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, an alarm PS0456, “ILLEGAL PARAMETER IN GRINDING” is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the alarm PS0456 is also issued.

5182	
	Axis number of dressing axis in Continuous feed surface grinding cycle(G78)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the axis number of dressing axis in Continuous feed surface grinding cycle(G78).

NOTE
The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, an alarm PS0456, “ILLEGAL PARAMETER IN GRINDING” is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the alarm PS0456 is also issued.

5183	Axis number of dressing axis in Intermittent feed surface grinding cycle(G79)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the axis number of dressing axis in Intermittent feed surface grinding cycle(G79).

NOTE
The axis number except for the cutting axis or grinding axis can be specified. When the axis number which is same to cutting axis or grinding axis is specified, an alarm PS0456, “ILLEGAL PARAMETER IN GRINDING” is issued at the time of execution. The Grinding Cycle is executed when this parameter value is 0 and address "L" is specified in NC program, the alarm PS0456 is also issued.

5184	In-position width for other than hole bottoms (regular)
5185	In-position width for other than hole bottoms (for retraction in peck drilling cycle)
5186	In-position width for other than hole bottoms (for shift in boring cycles (G76 and G87)
5187	In-position width for hole bottoms

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
Set an in-position width for drilling canned cycles for individual axes separately.

	#7	#6	#5	#4	#3	#2	#1	#0
5200						CRG		G84

[Input type] Parameter input
[Data type] Bit path

#0 G84 Method for specifying rigid tapping:
0: An M code specifying the rigid tapping mode is specified prior to the issue of the G84 (or G74) command. (See parameter No. 5210).
1: An M code specifying the rigid tapping mode is not used. (G84 cannot be used as a G code for the tapping cycle; G74 cannot be used for the reverse tapping cycle.)

#2 CRG Rigid mode when a rigid mode cancel command is specified (G80, 01 group G code, reset, etc.) :

0: Canceled after rigid tapping signal RGTAP is set to "0".
1: Canceled before rigid tapping signal RGTAP is set to "0".

	#7	#6	#5	#4	#3	#2	#1	#0
5202					CHR			

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#3 CHR Rigid tapping is:

0: Normal rigid tapping.
1: Interpolation type rigid tapping.

NOTE
1 The rigid tapping function and interpolation type rigid tapping function cannot be used together within a path.
2 Before the interpolation type rigid tapping function can be used, the Cs contour control function is required. If interpolation type rigid tapping is specified when the Cs contour control function is disabled, an alarm PS1223, "ILLEGAL SPINDLE SELECT" is issued.
3 The interpolation type rigid tapping function cannot be used in a path that has a spindle positioning axis. If interpolation type rigid tapping is specified for a path that has a spindle positioning axis, an alarm PS1223 is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
5203							HRM	HRG

[Input type] Parameter input
[Data type] Bit path

#0 HRG Rigid tapping by the manual handle is:

0: Disabled.
1: Enabled.

#1 HRM When the tapping axis moves in the negative direction during rigid tapping controlled by the manual handle, the direction in which the spindle rotates is determined as follows:

0: In G84 mode, the spindle rotates in a normal direction. In G74 mode, the spindle rotates in reverse.
1: In G84 mode, the spindle rotates in reverse. In G74 mode, the spindle rotates in a normal direction.

5241	Maximum spindle speed in rigid tapping (first gear)
5242	Maximum spindle speed in rigid tapping (second gear)
5243	Maximum spindle speed in rigid tapping (third gear)
5244	Maximum spindle speed in rigid tapping (fourth gear)

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 9999
Spindle position coder gear ratio
1 : 1 0 to 7400
1 : 2 0 to 9999
1 : 4 0 to 9999
1 : 8 0 to 9999
Each of these parameters is used to set a maximum spindle speed for each gear in rigid tapping.
Set the same value for both parameter No. 5241 and parameter No. 5243 for a one-stage gear system. For a two-stage gear system, set the same value as set in parameter No. 5242 in parameter No. 5243. Otherwise, alarm PS0200, “ILLEGAL S CODE COMMAND” will be issued. This applies to the M series.

	#7	#6	#5	#4	#3	#2	#1	#0
5400	SCR	XSC				D3R		

[Input type] Parameter input
[Data type] Bit path

#2 D3R When Reset is done by reset operation or reset signal from PMC, 3-dimensional coordinate system conversion mode and tilted working plane indexing mode is:
0: Canceled.
1: Not canceled.

NOTE
In case that workpiece setting error compensation is active, the 3-dimensional coordinate system conversion mode and tilted working plane indexing mode are canceled by a reset no matter what the setting of the parameter D3R.
And in case that tilted working plane indexing and tool center point control are used together, tilted working plane indexing mode is canceled by a reset no matter what the setting of the parameter D3R.

#6 XSC The setting of a scaling magnification (axis-by-axis scaling) is:
0: Disabled.
1: Enabled.

#7 SCR Scaling (G51) magnification unit:
0: 0.00001 times (1/100,000)
1: 0.001 times

	#7	#6	#5	#4	#3	#2	#1	#0
5401								SCLx

[Input type] Parameter input
[Data type] Bit axis

#0 SCLx Scaling on this axis:
0: Invalidated
1: Validated

	#7	#6	#5	#4	#3	#2	#1	#0
5402					DMK			

[Input type] Parameter input
[Data type] Bit path

#3 DMK The manual handle interruption screen is displayed:
0: Using the program coordinate system (post-conversion coordinate system).
1: Using the workpiece coordinate system (pre-conversion coordinate system).
This parameter is valid only in 3-dimensional coordinate conversion mode.

5411	Scaling (G51) magnification

[Input type] Setting input
[Data type] 2-word path
[Unit of data] 0.001 or 0.00001 times (Selected using bit 7 (SCR) of parameter No. 5400)
[Valid data range] 1to999999999
This parameter sets a scaling magnification when axis-by-axis scaling is disabled (with bit 6 (XSC) of parameter No. 5400 set to 0). If no scaling magnification (P) is specified in the program, the setting of this parameter is used as a scaling magnification.

NOTE
When bit 7 (SCR) of parameter No. 5400 is set to 1, the valid data range is 1 to 9999999.

5412	Rapid traverse rate for canned cycle for drilling in 3-dimensional coordinate conversion mode

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets a rapid traverse rate for canned cycle for drilling in the 3-dimensional coordinate conversion mode, the tilted working plane indexing mode, or the workpiece setting error compensation mode.

5421	Scaling magnification for each axis

[Input type] Setting input
[Data type] 2-word axis
[Unit of data] 0.001 or 0.00001 times (Selected using bit 7 (SCR) of parameter No. 5400)
[Valid data range] -999999999 to -1, 1 to 999999999

This parameter sets a scaling magnification for each axis when axis-by-axis scaling is enabled (with bit 6 (XSC) of parameter No. 5400 set to 1). For the first spindle to the third spindle (X-axis to Z-axis), the setting of this parameter is used as a scaling magnification if scaling magnifications (I, J, K) are not specified in the program.

NOTE
When bit 7 (SCR) of parameter No. 5400 is set to 1, the valid data ranges are -9999999 to -1 and 1 to 9999999.

	#7	#6	#5	#4	#3	#2	#1	#0
5431								MDL

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit path

#0 MDL The G60 code (single direction positioning) is:
0: One-shot G code (group 00).
1: Modal G code (group 01).

5440	Positioning direction and overrun distance in single direction positioning

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (A)
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the positioning direction and overrun distance in single direction positioning (G60) for each axis. The positioning direction is specified using a setting data sign, and the overrun distance using a value set here.
Overrun distance>0: The positioning direction is positive (+).
Overrun distance<0: The positioning direction is negative (-).
Overrun distance=0: Single direction positioning is not performed.

	#7	#6	#5	#4	#3	#2	#1	#0
5450						PLS		

[Input type] Parameter input
[Data type] Bit path

#2 PLS The polar coordinate interpolation shift function is:
0: Not used.
1: Used.
This enables machining using the workpiece coordinate system with a desired point which is not the center of the rotation axis set as the origin of the coordinate system in polar coordinate interpolation.

5460	Axis (linear axis) specification for polar coordinate interpolation

[Input type] Parameter input

[Data type] Byte path
[Valid data range] 1 to number of controlled axes
This parameter sets control axis numbers of linear axis to execute polar interpolation.

5461	Axis (rotation axis) specification for polar coordinate interpolation

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes
This parameter sets control axis numbers of rotation axis to execute polar interpolation.

5463	Automatic override tolerance ratio for polar coordinate interpolation

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 100
Typical setting: 90% (treated as 90% when set to 0)
Set the tolerance ratio of the fastest cutting feedrate to the speed of the rotation axis during automatic override of polar coordinate interpolation.

5464	Compensation for error on hypothetical axis of polar coordinate interpolation

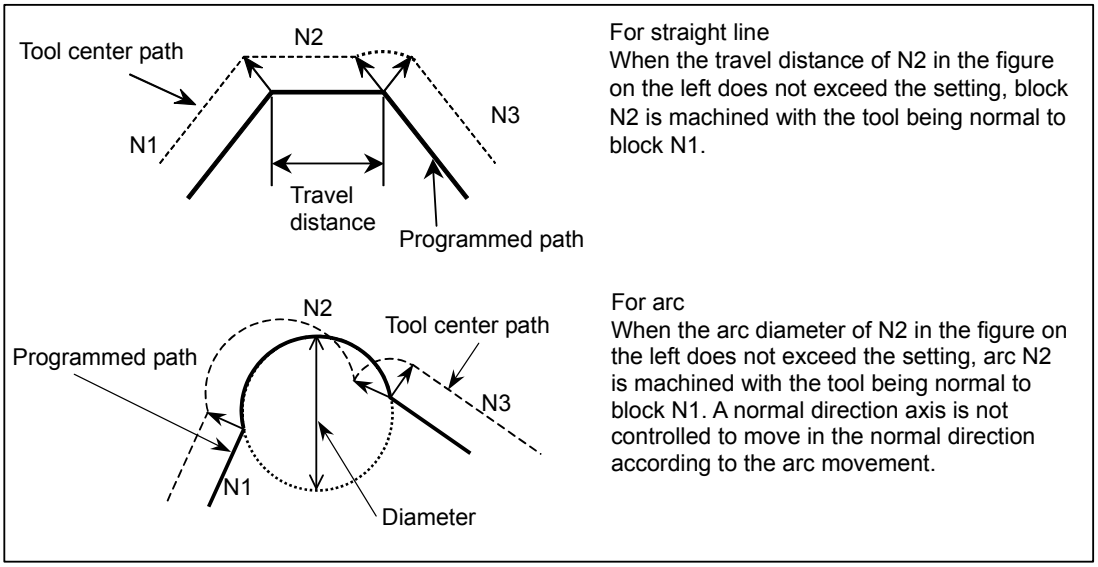
[Input type] Parameter input
[Data type] Byte path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(For IS-B, -999999.999 to +999999.999)
This parameter is used to set the error if the center of the rotation axis on which polar coordinate interpolation is performed is not on the X-axis.
If the setting of the parameter is 0, regular polar coordinate interpolation is performed.

5481	
	Feedrate of rotation of the normal direction controlled axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg/min
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
This parameter sets the feedrate of the movement along the normal direction controlled axis that is inserted at the start point of a block during normal direction control.

5483	
	Limit value of movement that is executed at the normal direction angle of a preceding block

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to standard parameter setting table (B)



	#7	#6	#5	#4	#3	#2	#1	#0
5630								SPN

[Input type] Parameter input
[Data type] Bit path

#0 SPN The amount of linear axis division (span value) in exponential interpolation is:
0: Specified with parameter No. 5643.
1: Specified using address K in a block containing G02.3/G03.3. When address K is not specified, the value set with parameter No. 5643 is used.

5641	Linear axis number subject to exponential interpolation

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes
This parameter sets the ordinal number, among the controlled axes, for the linear axis to which exponential interpolation is applied.

5642	Rotation axis number subject exponential interpolation

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of controlled axes
This parameter sets the ordinal number, among the controlled axes, for the rotation axis to which exponential interpolation is applied.

5643	Amount of linear axis division (span value) in exponential interpolation

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets an amount of linear axis division in exponential interpolation when bit 0 (SPN) of parameter No. 5630 is set to 0 or when address K is not specified.

	#7	#6	#5	#4	#3	#2	#1	#0
6000			SBM	HGO			MGO	G67
			SBM	HGO	V15		MGO	G67

[Input type] Parameter input
[Data type] Bit path

#0 G67 If the macro modal call cancel command (G67) is specified when the macro modal call mode (G66/G66.1) is not set:
0: Alarm PS1100, "CANCEL WITHOUT MODAL CALL" is issued.
1: The specification of G67 is ignored.

#1 MGO When a GOTO statement for specifying custom macro control is executed, a high-speed branch to 20 sequence numbers executed from the start of the program is:
0: A high-speed branch is not caused to n sequence numbers from the start of the executed program.
1: A high-speed branch is caused to n sequence numbers from the start of the program.

#3 V15 As system variable numbers for tool offset:
0 : The standard system variable numbers for the Series 16 are used.
1 : The same system variable numbers as those used for the Series 15 are used.
The tables indicate the system variables for tool offset numbers 1 to 999. The values for tool offset numbers 1 to 200 can be read from or assigned to the system variables in parentheses.
(1) Tool offset memory A

	System variable number	
	V15 = 0	V15 = 1
Wear offset value	#10001 to #10999 (#2001 to #2200)	#10001 to #10999 (#2001 to #2200)

(2) Tool offset memory B

	System variable number	
	V15 = 0	V15 = 1
Geometry offset value	#11001 to #11999 (#2201 to #2400)	#10001 to #10999 (#2001 to #2200)
Wear offset value	#10001 to #10999 (#2001 to #2200)	#11001 to #11999 (#2201 to #2400)

(3) Tool offset memory C

		System variable number	
		V15 = 0	V15 = 1
Tool length offset	Geometry offset value	#11001 to #11999 (#2201 to #2400)	#10001 to #10999 (#2001 to #2200)
	Wear offset value	#10001 to #10999 (#2001 to #2200)	#11001 to #11999 (#2201 to #2400)
Tool radius offset	Geometry offset value	#13001 to #13999	#12001 to #12999
	Wear offset value	#12001 to #12999	#13001 to #13999

#4 HGO When a GOTO statement for specifying custom macro control is executed, a branch to 30 sequence numbers just before the GOTO statement or to up to 10 sequence numbers saved by a sequence number search previously made with a GOTO statement is:
0: Not made at high speed.
1: Made at high speed.

#5 SBM Custom macro statement
0: Not stop the single block
1: Stops the single block

If you want to disable the single blocks in custom macro statements using system variable #3003, set this parameter to 0. If this parameter is set to 1, the single blocks in custom macro statements cannot be disabled using system variable #3003. To control single blocks in custom macro statements using system variable #3003, use bit 7 (SBV) of parameter No. 6000.

	#7	#6	#5	#4	#3	#2	#1	#0
6001		CCV	TCS	CRO	PV5		PRT	MIF

[Input type] Parameter input
[Data type] Bit path

#0 MIF The custom macro interface signals are based on:
0: Standard specification.
(The signals UI000 to UI015, UO000 to UO015, and UO100 to UO131 are used.)
1: Extended specification.
(The signals UI000 to UI031, UI100 to UI131, UI200 to UI231, UI300 to UI331, UO000 to UO031, UO100 to UO131, UO200 to UO231, and UO300 to UO331 are used.)

#1 PRT Reading zero when data is output using a DPRINT command
0: Outputs a space
1: Outputs no data

#3 PV5 Custom macro common variables:
0: #500 to #549 are output. (Note)
1: #100 to #149 and #500 to 549 are output. (Note)

NOTE
Output variables are as the table according to the combination of added options.

When the parameter PV5=0

		Option "Addition of custom macro common variables"	
		Non	Yes
Option "Embedded macro"	Non	#500 to #549	#500 to #999
	Yes	#500 to #549	#500 to #999

When the parameter PV5=1

		Option "Addition of custom macro common variables"	
		Non	Yes
Option "Embedded macro"	Non	#100 to #149 and #500 to #549	#100 to #199 and #500 to #999
	Yes	#100 to #149, #200 to #499 and #500 to #549	#100 to #199, #200 to #499 and #500 to #999

#4 CRO ISO code in BPRWT or DPRNT command
0: Outputs only "LF" after data is output
1: Outputs "LF" and "CR" after data is output

#5 TCS Custom macro (subprogram)
0: Not called using a T code
1: Called using a T code

#6 CCV Common variables #100 to #149(NOTE) cleared by power-off are:
0: Cleared to <null> by reset
1: Not cleared by reset

NOTE
Cleared variables are as the table according to the combination of added options.

		Option "Addition of custom macro common variables"	
		Non	Yes
Option "Embedded macro"	Non	#100 to #149	#100 to #199
	Yes	#100 to #149 and #200 to #499	#100 to #199 and #200 to #499

	#7	#6	#5	#4	#3	#2	#1	#0
6003			MSB	MPR	TSE	MIN		

[Input type] Parameter input
[Data type] Bit path

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#2 MIN Custom macro interrupt
0: Performed by interrupting an in-execution block (Custom macro interrupt type I)
1: Performed after an in-execution block is completed (Custom macro interrupt type II)

#3 TSE Custom macro interrupt signal UINT
0: Edge trigger method (Rising edge)
1: Status trigger method

#4 MPR Custom macro interrupt valid/invalid M code
0: M96/M97
1: M code set using parameters Nos. 6033 and 6034

#5 MSB Interrupt program
0: Uses a dedicated local variable (Macro-type interrupt)
1: Uses the same local variable as in the main program (Subprogram- type interrupt)

	#7	#6	#5	#4	#3	#2	#1	#0
6004						VHD		NAT
			D15					NAT

[Input type] Parameter input
[Data type] Bit path

#0 NAT The results of the custom macro functions ATAN (with 2 arguments) and ASIN are specified as follows:
0: The result of ATAN is 0 to 360.0.
The result of ASIN is 270.0 to 0 to 90.0.
1: The result of ATAN is -180.0 to 0 to 180.0.
The result of ASIN is -90.0 to 0 to 90.0.

#2 VHD With system variables #5121 to #5140:
0: The tool offset value (geometry offset value) in the block currently being executed is read. (This parameter is valid only when tool geometry/tool wear compensation memories are available.)
1: An interrupt travel distance based on manual handle interrupt is read.

#5 D15 When tool compensation memory C is used, for reading or writing tool offset values (for up to offset number 200) for D code (tool radius), the same system variables, #2401 through #2800, as Series 15 are:
0: Not used.
1: Used.
When bit 3 (V15) of parameter No. 6000 is set to 1

D code				
Compensation number	Geometry		Wear	
	Variable number	Variable name	Variable number	Variable name
1	#2401	[#_OFSDG[1]]	#2601	[#_OFSDW[1]]
2	#2402	[#_OFSDG[2]]	#2602	[#_OFSDW[2]]
3	#2403	[#_OFSDG[3]]	#2603	[#_OFSDW[3]]
:	:	:	:	:
199	#2599	[#_OFSDG[199]]	#2799	[#_OFSDW[199]]
200	#2600	[#_OFSDG[200]]	#2800	[#_OFSDW[200]]

	#7	#6	#5	#4	#3	#2	#1	#0
6005								SQC

[Input type] Parameter input
[Data type] Bit path

#0 SQC In the subprogram call function, a subprogram sequence number call is:
0: Not used.
1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
6007	SKM			CVA	MGE	BCS	SCS	DPG

[Input type] Parameter input
[Data type] Bit path

#0 DPG Specifies whether to allow G codes with a decimal point to be called.
0: Do not allow.
1: Allow.

#1 SCS Specifies whether to call subprograms with S codes.
0: Do not call with S codes.
1: Call with S codes.

#2 BCS Specifies whether to call subprograms with the second auxiliary function codes.
0: Do not call with the second auxiliary function codes.
1: Call with the second auxiliary function codes.

#3 MGE Specifies whether a G code modal call is made after movement or for each block.
0: Make a call for each block (equivalent to G66.1).
1: Make a call after movement (equivalent to G66).

#4 CVA The format for macro call arguments is specified as follows:

0: Arguments are passed in NC format without modifications.

1: Arguments are converted to macro format then passed.

[Example]

When G65 P_ X10 ; is specified, the value in local variable #24 in the calling program is set as follows:

Command	CVA=0	CVA=1
#24	0.01	0.01
ADP[#24]	10.0	0.01

NOTE
External operations are the same unless the ADP function is used.

#7 SKM After skip operation, with the workpiece coordinate system setting command (G92 for the M series or G50 for the T series) or select command (G54 to G59), the values of macro variables #100151 to #100200 (#5061 to #5080) holding the skip position:

0: Change. (The workpiece coordinate system at the time of reading is reflected.)

1: Do not change.

	#7	#6	#5	#4	#3	#2	#1	#0
6008	IJK	GMP		ISO			MCA	F16

[Input type] Parameter input

[Data type] Bit path

#0 F16 The precision of operation is based on:

0: New specification.

1: FS16*i* compatible specification.

#1 MCA A macro alarm specification based on system variable #3000 is selected as follows:

0: An alarm number obtained by adding 3000 to a value assigned to variable #3000 and the corresponding message are displayed. (A value from 0 to 200 can be assigned to variable #3000.)

1: A value assigned to variable #3000 and the corresponding message are displayed. (A value from 0 to 4095 can be assigned to variable #3000.)

[Example]

Execution of #3000=1 (ALARM MESSAGE);

When bit 1 (MCA) of parameter No. 6008 is set to 0:

The alarm screen displays "MC 3001 ALARM MESSAGE".

When bit 1 (MCA) of parameter No. 6008 is set to 1:

The alarm screen displays "MC0001 ALARM MESSAGE".

#4 ISO 0: When the EIA code is used, the bit patterns of codes specified instead of [,], #, *, =, ?, @, &, and _ are set in parameter No. 6010 to No. 6018.

1: When the ISO/ASCII code is used, the bit patters of codes specified instead of [,], #, *, =, ?, @, &, and _ are set in parameter No. 6010 to No. 6018.

#6 GMP The calling of M, S, T, a second auxiliary function code, or a particular code during the calling of a G code, and the calling of a G code during the calling of M, S, T, a second auxiliary function code, or particular code are:

0: Not allowed. (They are executed as an ordinary G, M, S, T, second auxiliary function code, and NC address.)

1: Allowed.

#7 IJK For addresses I, J, and K specified as arguments:
0: Argument specification I or II is automatically determined.
1: Argument specification I is always used.

Example
When K_J_I_ is specified:
- When this parameter is set to 0:
  Argument specification II is used and K=#6, J=#8, and I=#10 are specified.
- When this parameter is set to1:
  Argument specification I is used and I=#4, J=#5, and K=#6 are specified regardless of the specification order.
  (Argument specification II cannot be used.)

	#7	#6	#5	#4	#3	#2	#1	#0
6009								MSM

[Input type] Parameter input
[Data type] Bit path

#0 MSM When a Macro Call using M code or a Macro Call using M code (Specification of 3 Sets) are not at the beginning of the block:
0: Alarm PS0127, "DUPLICATE NC,MACRO STATEMENT" is issued.
1: The Macro Call using M code is executed. All addresses specified in the same block are used as arguments.

NOTE
1 When MSM is set to 1 and an M code specified for a macro call is not at the beginning of the block, argument specification II cannot be used.
2 When MSM is set to 1 and an M code specified for a macro call is not at the beginning of the block, the number of repetitions (L) cannot be used.

	#7	#6	#5	#4	#3	#2	#1	#0
6010	*7	*6	*5	*4	*3	*2	*1	*0
	#7	#6	#5	#4	#3	#2	#1	#0
6011	=7	=6	=5	=4	=3	=2	=1	=0
	#7	#6	#5	#4	#3	#2	#1	#0
6012	#7	#6	#5	#4	#3	#2	#1	#0
	#7	#6	#5	#4	#3	#2	#1	#0
6013	[7	[6	[5	[4	[3	[2	[1	[0
	#7	#6	#5	#4	#3	#2	#1	#0
6014	]7	]6	]5	]4	]3	]2	]1	]0
	#7	#6	#5	#4	#3	#2	#1	#0
6015	?7	?6	?5	?4	?3	?2	?1	?0
	#7	#6	#5	#4	#3	#2	#1	#0
6016	@7	@6	@5	@4	@3	@2	@1	@0
	#7	#6	#5	#4	#3	#2	#1	#0
6017	&7	&6	&5	&4	&3	&2	&1	&0
	#7	#6	#5	#4	#3	#2	#1	#0
6018	_7	_6	_5	_4	_3	_2	_1	_0

[Input type] Parameter input
[Data type] Bit path

*0 to *7 : The bit pattern of the EIA or ISO/ASCII code indicating * is set.
=0 to =7 : The bit pattern of the EIA or ISO/ASCII code indicating = is set.
#0 to #7 : The bit pattern of the EIA or ISO/ASCII code indicating # is set.
[0 to [7 : The bit pattern of the EIA or ISO/ASCII code indicating [is set.
]0 to]7 : The bit pattern of the EIA or ISO/ASCII code indicating] is set.
?0 to ?7 : The bit pattern of the EIA or ISO/ASCII code indicating ? is set.
@0 to @7 : The bit pattern of the EIA or ISO/ASCII code indicating @ is set.
&0 to &7 : The bit pattern of the EIA or ISO/ASCII code indicating & is set.
_0 to _7 : The bit pattern of the EIA or ISO/ASCII code indicating _ is set.
0: A corresponding bit is 0.
1: A corresponding bit is 1.

	#7	#6	#5	#4	#3	#2	#1	#0
6019	SFN		EDP		OFN	DPD		MCO
	SFN		EDP		OFN			MCO

[Input type] Parameter input
[Data type] Bit

#0 MCO When data is output, the decimal number value of the macro variable data is
0: Not output as a comment.
1: Output at the same time as a comment.

After the number, data, and the variable name of the macro variable are output when data output operation is performed the variable number and the value of the macro variable data in decimal number are output as a comment.

NOTE
1 Output data by this parameter is "Comment", and this is ignored at the time of reading.
2 Accuracy of the output data of the comment is up to 15 digits. The range of output data are nine digits above decimal point and eight digits below decimal point. "± OVER FLOW" is output instead of a value when the total digits number is more than 16 and the digit number above the decimal point is ten or more. When the number of digits below the decimal point becomes nine digits or more, the ninth place of the decimal point is rounded off and output. Moreover, the seventh place or the eighth place of the decimal point is rounded off and output when the total digits number is more than 16 and the digit number above decimal point is nine or eight.
3 The output becomes "EMPTY" when displayed, the macro variable data is "DATA EMPTY".

#2 DPD When argument D is specified for a macro call without a decimal point, the number of decimal places:

0: Is assumed to be 0.
[Example]
When G65PppppD1 is specified, #7=1.0000 is passed as the argument.

1: Depends on the increment system of the reference axis.
[Example]
When the increment system of the reference axis is IS-B and G65PppppD1 is specified, #7=0.0010 is passed as the argument.

#3 OFN The format of the name of a file output by the external output command (DPRNT or BPRNT) is:

0: PRNTxxxx.DAT (xxxx: 0000 to 9999).
1: MCR_PRNT.TXT (fixed).

#5 EDP Precision setting for macro relational operators is:

0: Disabled.
1: Enabled.

> **NOTE**
> Parameter No. 6100 is used to set the number of significant digits after the decimal point.

#7 SFN The format of the name of a file output by the external output command (DPRNT or BPRNT) is:

0: PRNTxxxx.DAT (xxxx: 0000 to 9999).
If the power is off and on, xxxx is reset to 0000.
1: PRNTxxxx.DAT (xxxx: 0000 to 9999).
If the power is off and on, xxxx is continued and the next number is applied.

> **NOTE**
> The setting value of bit 7 (SFN) is effective in case of that bit 3 (OFN) equals 0.

	#7	#6	#5	#4	#3	#2	#1	#0
6020						IFR		

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameters is set, the power must be turned off before operation is continued.

#2 IFR The custom macro interface signal R address is:

0: Disabled.
1: Enabled.

6030	M code to execute external device subprogram calls

[Input type] Setting input
[Data type] 2-word path
[Valid data range] 0 to 99999999

Set the M code to execute external device subprogram calls. When 0 is set, M198 is used. M01, M02, M30, M98, and M99 cannot be used to execute external device subprogram calls. When a negative number, 1, 2, 30, 98, or 99 is set for this parameter, M198 is used to execute external device subprogram calls.

6031	Start number of common variables to be protected among the common variables (#500 to #999)
6032	End number of common variables to be protected among the common variables (#500 to #999)

[Input type] Parameter input
[Data type] Word path
[Valid data range] 500 to 999
Among the common variables #500 to #999, the range of common variables specified by this parameter can be protected (by setting their attributes to read-only). If a write attempt (on the left side) is made, an alarm is issued.

NOTE
Set 0 in both parameter No. 6031 and No. 6032 not to protect common variables.

6033	M code that validates a custom macro interrupt
6034	M code that invalidates a custom macro interrupt

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 03 to 99999999 (excluding 30, 98 and 99)
These parameters can be used when bit 4 (MPR) of parameter No. 6003 is 1. M96 is used as a valid M code and M97 is used as an invalid M code when MPR is 0, irrespective of the state of this parameter.

6036	Number of custom macro variables common to tool path (for #100 to #199 (#499))

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 400
When the memory common to paths is used, this parameter sets the number of custom macro common variables to be shared (custom macro variables common to paths). Common variables #100 to #199 (up to #499 in a system with the embedded macro option) may be shared. Ensure that the maximum number of usable macro common variables is not exceeded.

Example
When 20 is set in parameter No. 6036
#100 to #119: Shared by all paths
#120 to #149: Used by each path independently

NOTE
1 To use up to #199, the option for adding custom macro common variables is required.
2 To use up to #499, the embedded macro option is required.
3 When 0 or a negative value is set, the memory common to paths is not used.
4 When the option for embedded macro is effective and the option for addition of custom macro common variables is not effective, #150 to #199 can not be used but this parameter should be set the number which includes #150 to #199.

6037	Number of custom macro variables common to tool path (for #500 to #999)

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 500

When the memory common to paths is used, this parameter sets the number of custom macro common variables to be shared (custom macro variables common to paths). Common variables #500 to #999 may be shared. Ensure that the maximum number of usable macro common variables is not exceeded.

Example
When 50 is set in parameter No. 6037
#500 to #549: Shared by all paths
#550 to #599: Used by each path independently

NOTE
1 To use up to #999, the option for adding custom macro common variables is required.
2 When 0 or a negative value is set, the memory common to paths is not used.

6038	Start G code used to call a custom macro

[Input type] Parameter input
[Data type] Word path
[Valid data range] -9999 to 9999

6039	Start program number of a custom macro called by G code

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 9999

6040	Number of G codes used to call custom macros

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 255

Set this parameter to define multiple custom macro calls using G codes at a time. With G codes as many as the value set in parameter No. 6040 starting with the G code set in parameter No. 6038, the custom macros of program numbers as many as the value set in parameter No. 6040 starting with the program number set in parameter No. 6039 can be called. Set 0 in parameter No. 6040 to disable this mode of calling.
If a negative value is set in parameter No. 6038, the modal call mode is entered. Whether the modal call is equivalent to G66 or G66.1 depends on bit 3 (MGE) of parameter No. 6007.

[Example] When parameter No. 6038 = 900, parameter No. 6039 = 1000, and parameter No. 6040 = 100 are set, a set of 100 custom macro calls (simple calls) is defined as follows:

G900 → O1000
G901 → O1001
G902 → O1002
:
G999 → O1099

When the setting of parameter No. 6038 is changed to -900, the same set of custom macro calls (modal calls) is defined.

NOTE
1 When the following conditions are satisfied, all calls using these parameters are disabled:
  1) When a value not within the specifiable range is set in each parameter
  2) (Value of parameter No. 6039 + value of parameter No. 6040 - 1) > 9999
2 The specification of a mixture of simple calls and modal calls is not allowed.
3 If a range of G codes set by these parameters duplicate G codes specified in parameters Nos. 6050 to 6059, the calls specified by parameters Nos. 6050 to 6059 are made preferentially.

6041	Start G code with a decimal point used to call a custom macro

[Input type] Parameter input
[Data type] Word path
[Valid data range] -999 to 999

6042	Start program number of a custom macro called by G code with a decimal point

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 9999

6043	Number of G codes with a decimal point used to call custom macros

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 255

Set this parameter to define multiple custom macro calls using G codes with a decimal point at a time. With G codes with a decimal point as many as the value set in parameter No. 6043 starting with the G code with a decimal point set in parameter No. 6041, the custom macros of program numbers as many as the value set in parameter No. 6043 starting with the program number set in parameter No. 6042 can be called. Set 0 in parameter No. 6043 to disable this mode of calling.
If a negative value is set in parameter No. 6041, the modal call mode is entered. Whether the modal call is equivalent to G66 or G66.1 depends on bit 3 (MGE) of parameter No. 6007.

[Example] When parameter No. 6041 = 900, parameter No. 6042 = 2000, and parameter No. 6043 = 100 are set, a set of 100 custom macro calls (simple calls) is defined as follows:
G90.0 → O2000
G90.1 → O2001
G90.2 → O2002
:
G99.9 → O2099
When the setting of parameter No. 6041 is changed to -900, the same set of custom macro calls (modal calls) is defined.

NOTE
1 When the following conditions are satisfied, all calls using these parameters are disabled:
  1) When a value not within the specifiable range is set in each parameter
  2) (Value of parameter No. 6042 + value of parameter No. 6043 - 1) > 9999
  3) When bit 0 (DPG) of parameter No. 6007 = 0 (to disable calls using G codes with a decimal point)
2 The specification of a mixture of simple calls and modal calls is not allowed.
3 If a range of G codes set by these parameters duplicate G codes specified in parameters Nos. 6060 to 6069, the calls specified by parameters Nos. 6060 to 6069 are made preferentially.

6044	Start M code used to call a subprogram

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999

6045	Start program number of a subprogram called by M code

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 9999

6046	Number of M codes used to call subprograms (number of subprograms called by M codes)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 32767

Set this parameter to define multiple subprogram calls using M codes at a time. With M codes as many as the value set in parameter No. 6046 starting with the M code set in parameter No. 6044, the subprograms of program numbers as many as the value set in parameter No. 6046 starting with the program number set in 6045 can be called. Set 0 in parameter No. 6046 to disable this mode of calling.

[Example] When parameter No. 6044 = 80000000, parameter No. 6045 = 3000, and parameter No. 6046 = 100 are set, a set of 100 subprogram calls is defined as follows:

M80000000 → O3000
M80000001 → O3001
M80000002 → O3002
:
M80000099 → O3099

NOTE
1 When the following conditions are satisfied, all calls using these parameters are disabled:
  1) When a value not within the specifiable range is set in each parameter
  2) (Value of parameter No. 6045 + value of parameter No. 6046 - 1) > 9999
2 If a range of M codes set by these parameters duplicate M codes specified in parameter No. 6071 to No. 6079, the calls specified by parameter No. 6071 to 6079 are made preferentially.

6047 **Start M code used to call a custom macro**

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999

6048 **Start program number of a custom macro called by M code**

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 9999

6049 **Number of M codes used to call custom macros (number of custom macros called by M codes)**

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 32767

Set this parameter to define multiple custom macro calls using M codes at a time. With M codes as many as the value set in parameter No. 6049 starting with the M code set in parameter No. 6047, the custom macros of program numbers as many as the value set in parameter No. 6049 starting with the program number set in parameter No. 6048 can be called. Set 0 in parameter No. 6049 to disable this mode of calling.

[Example] When parameter No. 6047 = 90000000, parameter No. 6048 = 4000, and parameter No. 6049 = 100 are set, a set of 100 custom macro calls (simple calls) is defined as follows:

M90000000 → O4000
M90000001 → O4001
M90000002 → O4002
:
M90000099 → O4099

NOTE
1 When the following conditions are satisfied, all calls using these parameters are disabled:
  1) When a value not within the specifiable range is set in each parameter
  2) (Value of parameter No. 6048 + value of parameter No. 6049 - 1) > 9999
2 If a range of M codes set by these parameters duplicate M codes specified in parameter No. 6080 through No. 6089, the calls specified by parameter No. 6080 through 6089 are made preferentially.

6050	G code that calls the custom macro of program number 9010
6051	G code that calls the custom macro of program number 9011
6052	G code that calls the custom macro of program number 9012
6053	G code that calls the custom macro of program number 9013
6054	G code that calls the custom macro of program number 9014
6055	G code that calls the custom macro of program number 9015
6056	G code that calls the custom macro of program number 9016
6057	G code that calls the custom macro of program number 9017
6058	G code that calls the custom macro of program number 9018
6059	G code that calls the custom macro of program number 9019

[Input type] Parameter input
[Data type] Word path
[Valid data range] (-9999 to 9999 : excluding 0, 5, 65, 66 and 67)
Set the G codes used to call the custom macros of program numbers 9010 to 9019. However, note that when a negative value is set in this parameter, it becomes a modal call. For example, if this parameter is set to -11, the modal call mode is entered by G11.
Whether the modal call is equivalent to G66 or G66.1 depends on bit 3 (MGE) of parameter No. 6007.

6060	G code with a decimal point used to call the custom macro of program number 9040
6061	G code with a decimal point used to call the custom macro of program number 9041
6062	G code with a decimal point used to call the custom macro of program number 9042
6063	G code with a decimal point used to call the custom macro of program number 9043
6064	G code with a decimal point used to call the custom macro of program number 9044
6065	G code with a decimal point used to call the custom macro of program number 9045
6066	G code with a decimal point used to call the custom macro of program number 9046
6067	G code with a decimal point used to call the custom macro of program number 9047
6068	G code with a decimal point used to call the custom macro of program number 9048

6069	G code with a decimal point used to call the custom macro of program number 9049

[Input type] Parameter input
[Data type] Word path
[Valid data range] -999 to 999

Set the G codes used to call the custom macros of program numbers 9040 to 9049. However, note that when a negative value is set in this parameter, it becomes a modal call. For example, if this parameter is set to -11, the modal call mode is entered by G11.
Whether the modal call is equivalent to G66 or G66.1 depends on bit 3 (MGE) of parameter No. 6007. Set G codes in the format Gm.n. The value expressed by (m×10+n) is set in the parameter. The values m and n must satisfy the following relationships: 0 ≤m ≤99, 0 ≤n ≤9.

NOTE
Parameter Nos. 6060 to 6069 are valid when bit 0 (DPG) of parameter No. 6007 is set to 1.

6071	M code used to call the subprogram of program number 9001
6072	M code used to call the subprogram of program number 9002
6073	M code used to call the subprogram of program number 9003
6074	M code used to call the subprogram of program number 9004
6075	M code used to call the subprogram of program number 9005
6076	M code used to call the subprogram of program number 9006
6077	M code used to call the subprogram of program number 9007
6078	M code used to call the subprogram of program number 9008
6079	M code used to call the subprogram of program number 9009

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999 (excluding 30, 98 and 99)

These parameters set the M codes that call the subprograms of program numbers 9001 to 9009.

NOTE
If the same M code is set in these parameters, the younger number is called preferentially. For example, if 100 is set in parameter No. 6071 and 6072, and programs O9001 and O9002 both exist, O9001 is called when M100 is specified.

6080	M code used to call the custom macro of program number 9020
6081	M code used to call the custom macro of program number 9021
6082	M code used to call the custom macro of program number 9022
6083	M code used to call the custom macro of program number 9023
6084	M code used to call the custom macro of program number 9024

6085	M code used to call the custom macro of program number 9025
6086	M code used to call the custom macro of program number 9026
6087	M code used to call the custom macro of program number 9027
6088	M code used to call the custom macro of program number 9028
6089	M code used to call the custom macro of program number 9029

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 3 to 99999999 (excluding 30, 98 and 99)
Set the M codes used to call the custom macros of program numbers 9020 to 9029. The simple call mode is set.

NOTE
1 If the same M code is set in these parameters, the younger number is called preferentially. For example, if 200 is set in parameter No. 6081 and No. 6082, and programs O9021 and O9022 both exist, O9021 is called when M200 is specified.
2 If the same M code is set in a parameters Nos. 6071 to 6079 used to call subprograms and in a parameters Nos. 6080 to 6089 used to call custom macros, a custom macro is called preferentially. For example, if 300 is set in parameters Nos. 6071 and 6081, and programs O9001 and O9021 both exist, O9021 is called when M300 is specified.

6090	ASCII code that calls the subprogram of program number 9004
6091	ASCII code that calls the subprogram of program number 9005

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 65(A:41H) to 90(Z:5AH)
These parameters set the ASCII codes that call subprograms in decimal.
The settable addresses are indicated below.

Address	Parameter setting value	T series	M series
A	65	O	O
B	66	O	O
D	68	X	O
F	70	O	O
H	72	O	O
I	73	O	O
J	74	O	O
K	75	O	O
L	76	O	O
M	77	O	O
P	80	O	O
Q	81	O	O
R	82	O	O
S	83	O	O
T	84	O	O
V	86	X	O

Address	Parameter setting value	T series	M series
X	88	X	O
Y	89	X	O
Z	90	X	O

NOTE
1 When address L is set, the number of repeats cannot be specified.
2 Set 0 when no subprogram is called.

6093 **Top address of custom macro interface signal R address (input signal)**

6094 **Top address of custom macro interface signal R address (output signal)**

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to maximum address (multiple of 4. 0, 4, 8, ...)
Set the top address of custom macro interface signal R address. 128 signals starting at the top address are allocated.

[Example]

Parameter	System variable number	Signal to use	Attribute
No.6093=1000	#1068 #1069 #1070 #1071	R1000 to R1003 R1004 to R1007 R1008 to R1011 R1012 to R1015	R
No.6094=1100	#1168 #1169 #1170 #1171	R1100 to R1103 R1104 to R1107 R1108 to R1111 R1112 to R1115	R/W

*) The R and R/W attributes of variables represent, respectively, read-only and read/write enabled.

NOTE
1 Each value specified with this parameter must be a multiple of 4 (0, 4, 8, ⋯). Otherwise, this function is disabled.
2 If a non-existent R address or system relay address is set, the corresponding system variable becomes invalid. The effective R address area varies depending on the PMC used and its memory. Be sure to specify a usable range by checking the specification of the PMC in use.
3 When setting the parameter, make sure that the input signal addresses do not overlap with the output signal addresses.

6100 **Precision setting for relational operators**

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 15

This parameter sets the number of digits after the decimal point in two values to be compared using the custom macro relational operator. The two values are rounded off to the specified number of digits before comparison.

NOTE
1 This function is enabled by setting bit 5 (EDP) of parameter No. 6019 to 1.
2 This function is disabled if parameter No. 6100 is set to a value out of the valid data range.

	#7	#6	#5	#4	#3	#2	#1	#0
6200	SKF	SRE	SLS	HSS			SK0	GSK

[Input type] Parameter input
[Data type] Bit path

#0 GSK As a skip signal, the skip signal SKIPP is:
0: Invalid.
1: Valid.

#1 SK0 This parameter specifies whether the skip signal is made valid under the state of the skip signal SKIP and the multistage skip signals SKIP2 to SKIP8.
0: Skip signal is valid when these signals are 1.
1: Skip signal is valid when these signals are 0.

#4 HSS 0: The skip function does not use high-speed skip signals while skip signals are input. (The conventional skip signal is used.)
1: The step skip function uses high-speed skip signals while skip signals are input.

#5 SLS 0: The multi-step skip function does not use high-speed skip signals while skip signals are input. (The conventional skip signal is used.)
1: The multi-step skip function uses high-speed skip signals while skip signals are input.

NOTE
The skip signals (SKIP and SKIP2 to SKIP8) are valid regardless of the setting of this parameter. They can also be disabled using bit 4 (IGX) of parameter No. 6201.
If you want to use high-speed skip signals when the multi-step skip function option is used, set this parameter to 1.

#6 SRE When a high-speed skip signal or high-speed measurement position arrival signal is used:
0: The signal is assumed to be input on the rising edge (contact open → close).
1: The signal is assumed to be input on the falling edge (contact close → open).

#7 SKF Dry run, override, and automatic acceleration/deceleration for G31 skip command
0: Disabled
1: Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
6201	SKPXE		CSE	IGX		TSE	SEB	

[Input type] Parameter input

[Data type] Bit path

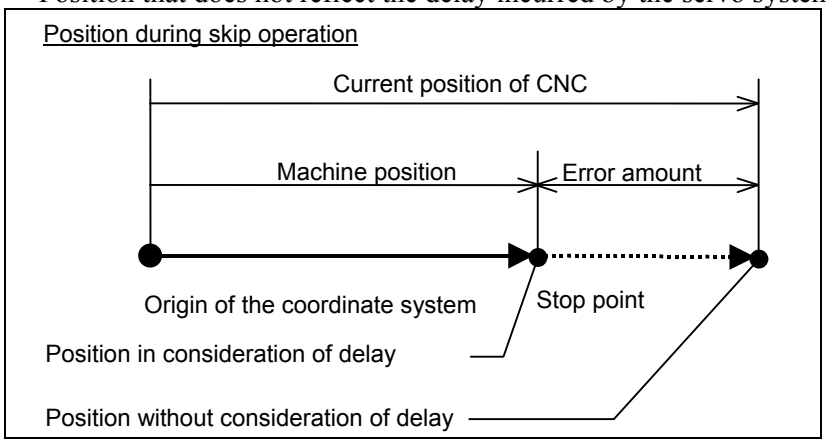
#1 SEB When a skip signal or measurement position arrival signal goes on while the skip function, or the automatic tool length measurement (M series) or automatic tool compensation (T series) is used, the accumulated pulses and positional deviation due to acceleration/deceleration are:

0: Ignored.
1: Considered and compensated.

The accumulated pulses and positional deviation due to actual acceleration/deceleration when the skip signal or measurement position arrival signal goes on are considered to obtain the position at which the signal is input.

#2 TSE When the torque limit skip function (G31 P98/99) is used, the skip position held in a system variable (#5061 to #5080 (#100151 to #100182 when 21-axes-or-more-system is used)) is:

0: Position that is offset considering the delay (positional deviation) incurred by the servo system.
1: Position that does not reflect the delay incurred by the servo system.



#4 IGX When the high-speed skip function is used, SKIP, SKIPP, and SKIP2 to SKIP8 are:

0: Enabled as skip signals.
1: Disabled as skip signals.

#5 CSE For the continuous high-speed skip command, high-speed skip signals are:

0: Effective at either a rising or falling edge (depending on the setting of bit 6 (SRE) of parameter No. 6200).
1: Effective at both the rising and falling edges.

#7 SKPXE For the skip function (G31), the skip signal SKIP is:

0: Enabled.
1: Disabled.

Whether the skip signals are enabled or disabled

Parameter	Bit 4 (IGX) of parameter No. 6201	Bit 0 (GSK) of parameter No. 6200	Bit 7 (SKPXE) of parameter No .6201	Skip signal SKIPP	Skip signal SKIP	Multistage skip signals SKIP2-SKIP8
Setting	0	0	0	Disabled	Enabled	Enabled
	0	1	0	Enabled	Enabled	Enabled
	0	0	1	Disabled	Disabled	Enabled
	0	1	1	Enabled	Disabled	Enabled
	1	0	0	Disabled	Disabled	Disabled
	1	1	0	Disabled	Disabled	Disabled
	1	0	1	Disabled	Disabled	Disabled
	1	1	1	Disabled	Disabled	Disabled

Bit 4 (IGX) of parameter No. 6201 is valid for the skip function using high-speed skip signals (when bit 4 (HSS) of parameter No. 6200 is set to 1) or for the multistage skip function using high-speed skip signals (when bit 5 (SLS) of parameter No. 6200 is set to 1).
To use multistage skip signals, the multistage skip function option is required.

	#7	#6	#5	#4	#3	#2	#1	#0
6202	1S8	1S7	1S6	1S5	1S4	1S3	1S2	1S1

[Input type] Parameter input
[Data type] Bit path

1S1 to 1S8 These parameters specify whether to enable or disable each high-speed skip signal when the G31 skip command is issued.
The following table shows the correspondence between the bits, input signals, and commands.
The settings of the bits have the following meaning :
0: The high-speed skip signal corresponding to a bit is disabled.
1: The high-speed skip signal corresponding to a bit is enabled.

Parameter	High-speed skip signals	Parameter	High-speed skip signals
1S1	HDI0	1S5	HDI4
1S2	HDI1	1S6	HDI5
1S3	HDI2	1S7	HDI6
1S4	HDI3	1S8	HDI7

NOTE
Do not specify the same signal simultaneously for different paths.

	#7	#6	#5	#4	#3	#2	#1	#0
6203	2S8	2S7	2S6	2S5	2S4	2S3	2S2	2S1
	#7	#6	#5	#4	#3	#2	#1	#0
6204	3S8	3S7	3S6	3S5	3S4	3S3	3S2	3S1
	#7	#6	#5	#4	#3	#2	#1	#0
6205	4S8	4S7	4S6	4S5	4S4	4S3	4S2	4S1
	#7	#6	#5	#4	#3	#2	#1	#0
6206	DS8	DS7	DS6	DS5	DS4	DS3	DS2	DS1

[Input type] Parameter input
[Data type] Bit path

1S1to1S8, 2S1to2S8, 3S1to3S8, 4S1to4S8, DS1toDS8

Specify which skip signal is enabled when the skip command (G31, or G31P1 to G31P4) and the dwell command (G04, G04Q1 to G04Q4) are issued with the multi-step skip function.

The following table shows the correspondence between the bits, input signals, and commands.

The setting of the bits have the following meaning :

0: The skip signal corresponding to a bit is invalid.

1: The skip signal corresponding to a bit is enabled.

Multi-step skip function

Command / Input signal	G31 G31P1 G04Q1	G31P2 G04Q2	G31P3 G04Q3	G31P4 G04Q4	G04
SKIP/HDI0	1S1	2S1	3S1	4S1	DS1
SKIP2/HDI1	1S2	2S2	3S2	4S2	DS2
SKIP3/HDI2	1S3	2S3	3S3	4S3	DS3
SKIP4/HDI3	1S4	2S4	3S4	4S4	DS4
SKIP5/HDI4	1S5	2S5	3S5	4S5	DS5
SKIP6/HDI5	1S6	2S6	3S6	4S6	DS6
SKIP7/HDI6	1S7	2S7	3S7	4S7	DS7
SKIP8/HDI7	1S8	2S8	3S8	4S8	DS8

NOTE
HDI0 to HDI7 are high-speed skip signals. Do not specify the same signal simultaneously for different paths.

When bit 0 (GSK) of parameter No. 6200 is set to 1, commands to be skipped can be selected by setting the following parameter:

Commands skipped by SKIPP signal <G006.6>

Parameter	Command skipped
When bit 0 (1S1) of parameter No. 6202 is set to 1	G31P1,G04Q1
When bit 0 (2S1) of parameter No. 6203 is set to 1	G31P2,G04Q2
When bit 0 (3S1) of parameter No. 6204 is set to 1	G31P3,G04Q3
When bit 0 (4S1) of parameter No. 6205 is set to 1	G31P4,G04Q4
When bit 0 (DS1) of parameter No. 6206 is set to 1	G04,G04Q1,G04Q2,G04Q3,G04Q4

	#7	#6	#5	#4	#3	#2	#1	#0
6207						SFN	SFP	

[Input type] Parameter input

[Data type] Bit path

#1 SFP The feedrate used when the skip function (G31) is being executed is:

0: Feedrate of a programmed F code.

1: Feedrate set in parameter No. 6281.

NOTE
For the multi-stage skip function and high-speed skip, see the description of bit 2 (SFN) of parameter No. 6207.

#2 SFN The feedrate used when the skip function based on high-speed skip signals (with bit 4 (HSS) of parameter No. 6200 set to 1) or the multi-skip function is being executed is:
0: Feedrate of a programmed F code.
1: Feedrate set in a parameter from parameter No. 6282 to No. 6285.

NOTE
For not the multistage skip function, but the skip function using no high-speed skip signals (when bit 4 (HSS) of parameter No. 6200 is set to 0), see the description of bit 1 (SFP) of parameter No. 6207.

	#7	#6	#5	#4	#3	#2	#1	#0
6208	9S8	9S7	9S6	9S5	9S4	9S3	9S2	9S1

[Input type] Parameter input
[Data type] Bit path

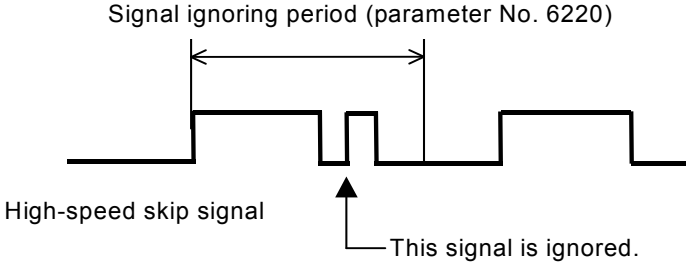
9S1 to 9S8 Specify which high-speed skip signal is enabled for the continuous high-speed skip command G31P90 or the EGB skip and the skip function for flexible synchronous control command G31.8.
The settings of each bit have the following meaning:
0: The high-speed skip signal corresponding to the bit is disabled.
1: The high-speed skip signal corresponding to the bit is enabled.
The bits correspond to signals as follows:

Parameter	High-speed skip signal	Parameter	High-speed skip signal
9S1	HDI0	9S5	HDI4
9S2	HDI1	9S6	HDI5
9S3	HDI2	9S7	HDI6
9S4	HDI3	9S8	HDI7

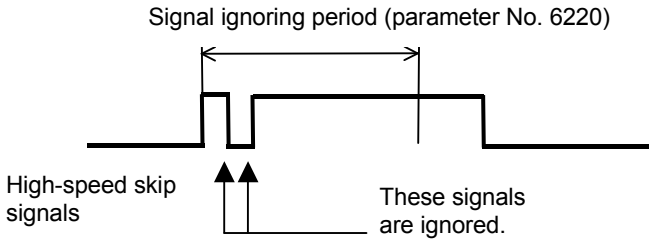
6220	Period during which skip signal input is ignored for the continuous high-speed skip function, EGB axis skip function and the skip function for flexible synchronous control

[Input type] Parameter input
[Data type] Byte path
[Unit of data] 8msec
[Valid data range] 3 to 127(× 8msec)

This parameter specifies the period from when a high-speed skip signal is input to when the next high-speed skip signal can be input for the continuous high-speed skip function and EGB axis skip function and the skip function for flexible synchronous control. This parameter is used to ignore chattering in skip signals.
If a value that falls outside the valid range is specified, the setting is assumed to be 24 msec.



When high-speed skip signals are used and bit 5 (CSE) of parameter No. 6201 is set to 1, signals are handled as follows:



6221	Torque limit dead zone time for a torque limit skip command

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] 2msec
[Valid data range] 0 to 65535
The torque limit skip arrival signal is ignored for a set period of time.
If G31P98 is specified, skip operation is not performed for a set period of time after the torque limit skip arrival signal is set to 1.
If G31P99 is specified, skip operation is not performed for a set period of time after the torque limit skip arrival signal is set to 1.
However, if a skip signal is input, skip operation is performed, regardless of the period of time set in this parameter.

6254	ε value on the X axis during automatic tool compensation (T series) ε value during automatic tool length measurement (M series) (for the XAE1 and GAE1 signals)

6255	ε value on the Z axis during automatic tool compensation (T series) ε value during automatic tool length measurement (M series) (for the XAE2 and GAE2 signals)

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] mm, inch, deg (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
These parameters set the relevant ε value during automatic tool compensation (T series) or automatic tool length measurement (M series).

NOTE
1 For the M series, when the setting of parameter No. 6252 or 6253 is 0, the setting of parameter No. 6251 is used.
2 Set a radius value regardless of whether diameter or radius programming is specified.

6281	Feedrate for the skip function (G31)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets a feedrate for the skip function (G31). This parameter is valid when bit 1 (SFP) of parameter No. 6207 is set to 1.

NOTE
For the multi-stage skip function and high-speed skip, see the description of parameter No. 6282 to No. 6285.

6282	Feedrate for the skip function (G31, G31 P1)
6283	Feedrate for the skip function (G31 P2)
6284	Feedrate for the skip function (G31 P3)
6285	Feedrate for the skip function (G31 P4)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Each of these parameters sets a feedrate for each skip function G code. These parameters are valid when bit 2 (SFN) of parameter No. 6207 is set to 1.

6287	Positional deviation limit in torque limit skip

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 327670
This parameter sets a positional deviation limit for each axis imposed when torque limit skip is specified. When the actual positional deviation exceeds the positional deviation limit, the alarm SV0004, “EXCESS ERROR (G31)” is issued and an immediate stop takes place.

	#7	#6	#5	#4	#3	#2	#1	#0
6400	MG4	MGO	RVN	HMP	MC8	MC5	FWD	RPO

[Input type] Parameter input
[Data type] Bit path

#0 RPO With the manual handle retrace function, the rapid traverse rate is clamped, assuming that:
0: An override of 10% is used.
1: An override of 100% is used.

#1 FWD With the manual handle retrace function, program execution can be performed:
0: In both forward and backward directions.
1: In the forward direction only. Execution in the backward direction is not permitted.

#2 MC5
#3 MC8 These parameters set the number of M code groups and the number of M codes per group. (See explanations of parameters Nos. 6411 to 6490.)

MC5	MC8	M code group setting
0	0	Standard (20 groups of four)
1	0	16 groups of five
0	1	10 groups of eight

When 16 groups of five are used, the meanings of parameters are changed as follows:
Group A No. 6411(1) to No. 6415(5)
Group B No. 6416(1) to No. 6420(5)
:
Group P No. 6486(1) to No. 6490(5)
When 10 groups of eight are used, they are changed as follows:
Group A No. 6411(1) to No. 6418(8)
Group B No. 6419(1) to No. 6426(8)
:
Group J No. 6483(1) to No. 6490(8)

#4 HMP When inversion or backward movement is inhibited in other paths:
0: Inversion or backward movement is not inhibited for the currently executed path.
1: Inversion or backward movement is inhibited also for the currently executed path.

#5 RVN When the manual handle retrace function is used, M codes other than grouped M codes:
0: Do not disable backward movement.
1: Disable backward movement.

When this parameter is set to 1, M codes other than grouped M codes disable backward movement in general. Exceptionally, however, the following M codes allow backward movement:
1. Subprogram call based on M98/M99
2. Subprogram call based on an M code
3. Macro call based on an M code
4. Waiting M code
5. M0

#6 MGO When the manual handle retrace function is used, handle pulses during execution of a G code related to measurement are:
0: Valid.
1: Invalid. A speed with an override of 100% is used for execution at all times.

#7 MG4 In the manual handle retrace function, for blocks for which multi-step skip G04 is enabled (when the multi-step skip software option is used, and the settings of parameters Nos. 6202 to 6206 are valid):
0: Backward movement is not prohibited.
1: Backward movement is prohibited.

	#7	#6	#5	#4	#3	#2	#1	#0
6401	STO	HST				CHS		

[Input type] Parameter input
[Data type] Bit path

#2 CHS In manual handle retrace:
0: The status is displayed if the following conditions are all satisfied:
(1) The manual handle retrace software option is used.
(2) Bit 6 (HST) of parameter No. 6401, which specifies whether to enable or disable status display, is set to 1.
(3) Check mode output signal MMMOD<Fn091.3> is set to 1.
1: The status is displayed if the following conditions are all satisfied:
(1) The manual handle retrace software option is used.
(2) Bit 6 (HST) of parameter No. 6401, which specifies whether to enable or disable status display, is set to 1.

(3) Cycle start lamp signal STL<Fn000.5> is set to 1.
(4) Check mode input signal MMOD<Gn067.2> is set to 1.
(5) Handle input signal MCHK<Gn067.3> is set to 1 in the check mode.

#6 HST When the manual handle retrace function is used, the time display field on the status display line of the CNC screen:
0: Does not display status.
1: Displays status.

#7 STO In the manual handle retrace function, the timing for outputting an S code and T code during backward movement is:
0: Different from the timing during forward movement:
1: The same as during forward movement.

	#7	#6	#5	#4	#3	#2	#1	#0
6403				HRE	HRD	HRC	HRB	HRA

[Input type] Parameter input
[Data type] Bit path

#0 HRA In rigid tapping and thread cutting, with the manual handle retrace function, program execution in the forward direction:
0: Cannot be performed.
1: Can be performed.

#1 HRB In PMC axis control, with the manual handle retrace, program execution in the forward direction:
0: Cannot be performed.
1: Can be performed.

#2 HRC During orientation operation according to G00 for a Cs contour control axis, with the manual handle retrace function, program execution in the backward direction:
0: Cannot be performed.
1: Can be performed.

#3 HRD In polygon machining with two spindles, with the manual handle retrace function, program execution in the backward direction:
0: Cannot be performed.
1: Can be performed.

#4 HRE In balanced cutting, with the manual handle retrace function, program execution in the backward direction:
0: Cannot be performed.
1: Can be performed.

6405	Override value (equivalence) for clamping the rapid traverse rate used with the manual handle retrace function

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 0 to 100

This parameter sets an override value (equivalence) for clamping the rapid traverse rate used with the manual handle retrace function. If a value greater than 100 is set in parameter No. 6405, the rapid traverse rate is clamped to an override of 100%. This function is invalid if 0 is set in parameter No. 6405. In this case, the setting of bit 0 (RPO) of parameter No. 6400 is used.

6410	Travel distance per pulse generated from the manual pulse generator

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 0 to 100

Set the travel distance per pulse generated from the manual pulse generator in terms of the override value.
The distance traveled by the machine when the manual handle is actually turned can be found by the following expression:
[Specified speed] × [Handle magnification] × ([Setting of this parameter]/100) × (8/60000) (mm or inch)

[Example] When a specified feedrate is 30mm/min, the manual handle magnification is 100, and parameter No. 6410 is set to 1, the travel distance per pulse generated from the manual pulse generator is calculated as follows:
[Travel distance per pulse]=30[mm/min] × 100 × (1/100) × (8/60000)[min]= 0.004mm

6411	M code of group A in manual handle retrace (1)
to	to
6414	M code of group A in manual handle retrace (4)
6415	M code of group B in manual handle retrace (1)
to	to
6418	M code of group B in manual handle retrace (4)
6419	M code of group C in manual handle retrace (1)
to	to
6422	M code of group C in manual handle retrace (4)
6423	M code of group D in manual handle retrace (1)
to	to
6426	M code of group D in manual handle retrace (4)
6427	M code of group E in manual handle retrace (1)
to	to
6430	M code of group E in manual handle retrace (4)
6431	M code of group F in manual handle retrace (1)
to	to
6434	M code of group F in manual handle retrace (4)
6435	M code of group G in manual handle retrace (1)
to	to
6438	M code of group G in manual handle retrace (4)
6439	M code of group H in manual handle retrace (1)
to	to
6442	M code of group H in manual handle retrace (4)

6443	M code of group I in manual handle retrace (1)
to	to
6446	M code of group I in manual handle retrace (4)
6447	M code of group J in manual handle retrace (1)
to	to
6450	M code of group J in manual handle retrace (4)
6451	M code of group K in manual handle retrace (1)
to	to
6454	M code of group K in manual handle retrace (4)
6455	M code of group L in manual handle retrace (1)
to	to
6458	M code of group L in manual handle retrace (4)
6459	M code of group M in manual handle retrace (1)
to	to
6462	M code of group M in manual handle retrace (4)
6463	M code of group N in manual handle retrace (1)
to	to
6466	M code of group N in manual handle retrace (4)
6467	M code of group O in manual handle retrace (1)
to	to
6470	M code of group O in manual handle retrace (4)
6471	M code of group P in manual handle retrace (1)
to	to
6474	M code of group P in manual handle retrace (4)
6475	M code of group Q in manual handle retrace (1)
to	to
6478	M code of group Q in manual handle retrace (4)
6479	M code of group R in manual handle retrace (1)
to	to
6482	M code of group R in manual handle retrace (4)
6483	M code of group S in manual handle retrace (1)
to	to
6486	M code of group S in manual handle retrace (4)
6487	M code of group T in manual handle retrace (1)
to	to
6490	M code of group T in manual handle retrace (4)

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 9999

Set a group of M codes output during backward movement.
For backward movement for an M code, the modal M code in the same group set by the parameter is output.
The first M code in each group is set as the default.
When the number of M codes in a group is 3 or less, set the parameter corresponding to an unused M code to 0.
For backward movement for "M0", "M0" is output regardless of which M code is set for the parameter. 0 set in the parameter is ignored.

For an M code which is not set in any group by any of the above parameters, the M code for forward movement is output.
With these parameters, an M code in the same group can be output in backward movement only when the M code is the first M code in each block. When a block contains two or more M codes, the same M codes as output in forward movement are output as a second M code and up.

NOTE
The above explanation of M code groups applies to the standard settings. The number of M codes in each group and the number of M code groups vary depending on the settings of bits 2 (MC5) and 3 (MC8) of parameter No. 6400.

6581	RGB value of color palette 1
to	to
6595	RGB value of color palette 15

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 to 151515
Each of these parameters sets the RGB value of each color palette by specifying a 6-digit number as described below.
rrggbb: 6-digit number (rr: red data, gg: green data, bb: blue data)
The valid data range of each color is 0 to 15 (same as the tone levels on the color setting screen). When a number equal to or greater than 16 is specified, the specification of 15 is assumed.
[Example] When the tone level of a color is: red:1 green:2, blue:3, set 10203 in the parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
6700								PCM

[Input type] Parameter input
[Data type] Bit path

#0 PCM M code that counts the total number of machined parts and the number of machined parts
0: M02, or M30, or an M code specified by parameter No. 6710
1: Only M code specified by parameter No. 6710

6710	M code that counts the number of machined parts

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 999999999
The total number of machined parts and the number of machined parts are counted (+1) when the M code set is executed.

NOTE
The setting of 0 is invalid (no count operation is performed with M00.) Moreover, M98, M99, M198 (external device subprogram calling), and M codes used for subprogram calling and macro calling cannot be set as M codes for count-up operation. (Even when such an M code is set, count-up operation is not performed, ignoring the M code.)

6711	Number of machined parts

[Input type] Setting input
[Data type] 2-word path
[Valid data range] 0 to 999999999
The number of machined parts is counted (+1) together with the total number of machined parts when the M02, M30, or a M code specified by parameter No. 6710 is executed.

NOTE
The number of parts is not counted for M02, M30 when bit 0 (PCM) of parameter No. 6700 is set to 1.

6712	Total number of machined parts

[Input type] Setting input
[Data type] 2-word path
[Valid data range] 0 to 999999999
This parameter sets the total number of machined parts.
The total number of machined parts is counted (+1) when M02, M30, or an M code specified by parameter No. 6710 is executed.

NOTE
The number of parts is not counted for M02, M30, when bit 0 (PCM) of parameter No. 6700 is set to 1.

6713	Number of required parts

[Input type] Setting input
[Data type] 2-word path
[Valid data range] 0 to 999999999
This parameter sets the number of required machined parts.
Required parts finish signal PRTSF <F0062.7> is output to PMC when the number of machined parts reaches the number of required parts. The number of parts is regarded as infinity when the number of required parts is zero. The PRTSF signal is then not output.

6750	Integrated value of power-on period

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min
[Valid data range] 0 to 999999999
This parameter displays the integrated value of power-on period.

6751	Operation time (integrated value of time during automatic operation) 1

[Input type] Setting input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 59999
For details, see the description of parameter No. 6752.

6752	Operation time (integrated value of time during automatic operation) 2

[Input type] Setting input
[Data type] 2-word path
[Unit of data] min
[Valid data range] 0 to 999999999
This parameter displays the integrated value of time during automatic operation (neither stop nor hold time included).
The actual time accumulated during operation is the sum of this parameters Nos. 6751 and 6752.

6753	Integrated value of cutting time 1

[Input type] Setting input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 59999
For details, see the description of parameter No. 6754.

6754	Integrated value of cutting time 2

[Input type] Setting input
[Data type] 2-word path
[Unit of data] min
[Valid data range] 0 to 999999999
This parameter displays the integrated value of a cutting time that is performed in cutting feed such as linear interpolation (G01) and circular interpolation (G02 or G03).
The actual time accumulated during cutting is the sum of this parameters Nos. 6753 and 6754.

	#7	#6	#5	#4	#3	#2	#1	#0
6800	M6T	IGI	SNG	GRS	SIG	LTM	GS2	GS1

[Input type] Parameter input
[Data type] Bit path

#0 GS1

#1 GS2 For the maximum number of groups set in parameter No. 6813, up to four tools can be registered per group. The combination of the number of registrable groups and the number of tools per group can be changed by setting GS1 and GS2.

GS2	GS1	Number of groups	Number of tools
0	0	1/8 of maximum number of groups (No. 6813)	32
0	1	1/4 of maximum number of groups (No. 6813)	16
1	0	1/2 of maximum number of groups (No. 6813)	8
1	1	Maximum number of groups (No. 6813)	4

NOTE
After changing these parameters, set data again by using G10 L3 ;(registration after deletion of data of all groups).

#2 LTM The tool life count is specified by:
0: Count.
1: Duration.

NOTE
After changing this parameter, set data again by using G10 L3 ;(registration after deletion of data of all groups).

#3 SIG When a tool is skipped by a signals TL01 to TL512 <Gn047.0 to Gn048.1>, the group number is:
0: Not input by the tool group number selection signals.
1: Input by the tool group number selection signals.

NOTE
When this parameter is set to 0, a tool of the currently used group is skipped.

#4 GRS When the tool change reset signal TLRST <Gn048.7> is input:
0: If the life of the group specified by the tool group number selection signals TL01 to TL512 <Gn047.0 to Gn048.1> has expired, the execution data of the group is cleared.
1: The execution data of all registered groups is cleared.
If this parameter is set to 1, the execution data of all registered groups is cleared also when the clear operation to clear execution data is performed on the tool life management list screen.

#5 SNG When the tool skip signal TLSKP <Gn048.5> is input while a tool not controlled by the tool life management function is being used:
0: A tool of the most recently used group or a specified group (bit 3 (SIG) of parameter No. 6800) is skipped.
1: The tool skip signal is ignored.

#6 IGI Tool back numbers are:
0: Not ignored.
1: Ignored.

#7 M6T A T code specified in the same block as M06 is:
0: Assumed to be a back number.
1: Assumed to be a command specifying the next tool group.

	#7	#6	#5	#4	#3	#2	#1	#0
6801	M6E				EMD	LVF	TSM	
	M6E				EMD	LVF		

[Input type] Parameter input
[Data type] Bit path

#1 TSM In the tool life management function, life counting is performed as follows when more than one offset is specified:
0: Counting is performed for each tool number.
1: Counting is performed for each tool.

#2 LVF When the life value is counted by duration in the tool management function or tool life management function, tool life count override signals *TLV0 to *TLV9 <Gn049.0 to Gn050.1> are:
0: Not used.
1: Used.

#3 EMD In the tool life management function, the mark "*" indicating that the life has expired is displayed when:

0: The next tool is used.
1: The life has just expired.

> **NOTE**
> If this parameter is set to 0, the "@" mark (indicating that the tool is in use) is kept displayed unless the next tool whose life has not expired is used. If this parameter is set to 1, marks are displayed in different ways depending on the life count type.
> If the life count type is the duration specification type, the "*" mark (indicating that the life has expired) appears when the life has expired. If the life count type is the count specification type, one count is not assumed until the end of the program (M02, M30, and so on). Therefore, even when the life value and the tool life counter value match, the "*" mark (life has expired) does not appear. The "*" mark (life has expired) appears when the tool is used again by a tool group command (T code) or tool change command (M06) issued after the CNC is reset.

#7 M6E When a T code is specified in the same block as M06:

0: The T code is treated as a back number or the group number to be selected next. Which number is assumed depends on the setting of bit 7 (M6T) of parameter No. 6800.
1: Life counting for the tool group starts immediately.

	#7	#6	#5	#4	#3	#2	#1	#0
6802	RMT	TSK	TGN	ARL	GRP	E17	TCO	T99

[Input type] Parameter input
[Data type] Bit path

#0 T99 When M99 of the main program is executed, and there is a the life was expired tool group:

0: The tool change signal TLCH <Fn064.0> is not output.
1: TLCH is output, and the automatic operation becomes a stopped state..

If the life count is specified by use count and this parameter 1, TLCH is output and the automatic operation becomes a stopped state if the life of at least one tool group has expired when the M99 command is specified.
If the life count type is the duration specification type, the automatic operation becomes a stopped state if the life of at least one tool group has expired when the M99 command is specified.

M

If the life count is specified by use count, after the M99 command is specified, a tool group command (T code) selects, from a specified group, a tool whose life has not expired, and the next tool change command (M06) increments the tool life counter by one.

T

If the life count is specified by use count, when a tool group command (T code) is specified after the M99 command is specified, a tool whose life has not expired is selected from a specified group, and the tool life counter is incremented by one.
When the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1), the same specifications as for the M series apply.

#1 TCO

#2 E17 Specifies whether to allow the FOCAS2 or PMC window function to write tool information of a group being used or a group to be used next during automatic operation (the automatic operation signal OP is set to 1).

Condition			Bit 1 (TCO) of parameter No. 6802		
			0	1	
				Bit 2 (E17) of parameter No. 6802	
				1	0
During automatic operation	Group being used or to be used next	Tool being used	×	□	○
		Tool not being used	×	○	○
	Group neither being used nor to be used next		○	○	○
Not during automatic operation			○	○	○

○: Tool information can be written from FOCAS2 and PMC window.
×: Tool information cannot be written from FOCAS2 and PMC window.
When an attempt is made to write tool information from PMC window, completion code 13 (REJECT ALARM) is returned.
□: Tool information cannot be cleared.

NOTE
When tool information of a tool being used (marked with "@") in the group being used or to be used next or tool information of the most recently used tool (marked with "@") in a group that is neither the group being used nor the group to be used next is cleared, the life counter is reset to 0.
It is possible to modify tool information of a tool in the group to be used next. However, because tool selection is already completed, the selected tool does not change even when the tool information is modified.
This parameter has no influence on modifications to tool information by edit operations from the tool life management screen.

#3 GRP Management data of tool life arrival notice signal TLCHB <Fn064.3> is:
0: Managed using the remaining life value set in parameter No. 6844 and 6845.
1: Managed using the remaining life value set in tool life management data.

NOTE
When the tool life arrival notice signal function is used, bit 4 (LFB) of parameter No. 6805 must be set to 1 to enable the tool life management B function.

#4 ARL Tool life arrival notice signal TLCHB <Fn064.3> of tool life management is:
0: Output for each tool.
1: Output for the last tool of a group.
This parameter is valid only when bit 3 (GRP) of parameter No. 6802 is set to 1.

#5 TGN In the tool life management function, the optional group number function is:
0: Not used.
1: Used.

> **NOTE**
> When the optional group number function is used, bit 4 (LFB) of parameter No. 6805 must be set to 1 to enable the tool life management B function.
> In lathe systems, the optional group number function can be used if the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1).

#6 TSK If the count type in tool life management is the duration type, then when the last tool of a group is skipped by a signal:
0: The count value for the last tool equals the life value.
1: The count value for the last tool remains unchanged.

#7 RMT Tool life arrival notice signal TLCHB <Fn064.3> is turned "1" and 0 as follows:
0: The signal is turned "1" if the remaining life value (the life value minus the life counter value) is smaller than or equal to the remaining life setting. The signal is turned 0 if the remaining life value (the life value minus the life counter value) is greater than the remaining life setting.
1: The signal is turned "1" if the remaining life value (the life value minus the life counter value) is equal to the remaining life setting. The signal is turned "0" if the remaining life value (the life value minus the life counter value) is not equal to the remaining life setting.

> **NOTE**
> When using the life count override feature, set bit 7 (RMT) of parameter No. 6802 to 0. When the life count is specified by duration, the unit used for determining the result of comparison between the remaining life and the remaining life setting varies depending on the life count interval (bit 0 (FCO) of parameter No. 6805). If the life is counted every second, the comparison is made in units of 1 minute; if the life is counted every 0.1 second, the comparison is made in units of 0.1 minute.

	#7	#6	#5	#4	#3	#2	#1	#0
6803	CTB							

[Input type] Parameter input
[Data type] Bit path

 # 7 CTB Whether to turn the tool life arrival notice signal TLCHB <Fn064.3> of tool life management off is determined when life counting starts. An additional turn-off condition is:
0: Not added.
1: Added.

The tool life arrival notice signal is turned off when one of the following operations is performed for the currently used group:

- Clears the execution data on the tool life management list screen.
- Deletes all tool group data at a time, adds a tool number, or deletes tool data on the tool life management edit screen.
- Clears the execution data by the tool change reset signal TLRST <Gn048.7>.
- Registers, changes, or deletes all tool life management group data by the G10 command.
- Executes the FOCAS2 cnc_clrcntinfo function (which clears the tool life counter or tool information).
- Replaces a tool with a tool of which life is not managed by the M06 command.

NOTE
When this parameter is set, the power must be turned off before operation is continued.

	#7	#6	#5	#4	#3	#2	#1	#0
6804		LFI				ETE	TCI	

[Input type] Parameter input
[Data type] Bit path

#1 TCI During automatic operation (the automatic operation signal OP<Fn000.7> is “1”), editing of tool life data is:
0: Disabled.
1: Enabled.

NOTE
When this parameter is set to 1, tool life data can be edited even during automatic operation (the OP is 1). If the target group for editing is the group being used or the group to be used next, however, only presetting of the life counter is permitted, and other data cannot be modified.

#2 ETE In the tool life management screen, the mark of the tool at the life was expired of the final tool in the group :
0: depends on setting bit 3 (EMD) of parameter No. 6801.
1: is "*" mark.
If bit 2 (ETE) of parameter No. 6804 is set to 1, when the life counter of the final tool in the group becomes equal to the life value, display mark "*" in the final tool of the tool life management screen.
When tool change signal TLCH <Fn064.0> is “1”, the state of the life was expired of the tool can be read by reading tool information on the final tool in FOCAS2 or the PMC window.

#6 LFI In tool life management, counting of the life of a selected tool is:
0: Enabled.
1: Enabled or disabled according to the status of tool life counting disable signal LFCIV <Gn048.2>.

	#7	#6	#5	#4	#3	#2	#1	#0
6805	TAD	TRU	TRS	LFB			FGL	FCO

[Input type] Parameter input
[Data type] Bit path

#0 FCO If the life count type is the duration specification type, the life is counted as follows:
0: Every second.
1: Every 0.1 second.
According to the setting of this parameter, the increment system of life values and tool life counter values displayed on the tool life management screen is set as follows:

Parameter FCO	0	1
Increment system for display and setting of life values and life counter values	1-minute increments	0.1-minute increments

NOTE
After changing the setting of this parameter, set data again by using G10L3;(registration after deletion of data of all groups).

#1 FGL If the life count type is the duration specification type, life data registered by G10 is:
0: In minute increments.
1: In 0.1-second increments.

#4 LFB The tool life management B function is:
0: Disabled.
1: Enabled.
When the tool life management B function is enabled, the following functions can be used:
<1> Tool life value extension (count specification: 99999999 times, duration specification: 100000 minutes)
<2> Optional group number function
<3> Tool life arrival notice function
In lathe systems, if the tool change type is the ATC type (bit 3 (TCT) of parameter No. 5040 = 1), the optional group number function can be used.

#5 TRS Tool change reset signal TLRST <Gn048.7> is valid when reset signal RST <Fn001.1> is not "1" and:
0: The reset state (automatic operation signal OP is "0") is observed.
1: The reset state (automatic operation signal OP <Fn000.7> is "0"), automatic operation stop state (cycle start lamp signal STL <Fn000.5> and feed hold lamp signal SPL <Fn000.4> are "0" and OP is "1"), or the automatic operation pause state (STL is "0" and SPL is "1") is observed. The TLRST signal, however, is invalid when the automatic operation stop state, automatic operation pause state, and automatic operation start state (STL is "1") is observed during execution of a data setting command (G10L3).

#6 TRU When the life count type is the duration specification type, and the life is counted every second (bit 0 (FCO) of parameter No. 6805 is set to 0):
0: Cutting time less than one second is discarded and is not counted.
1: Cutting time less than one second is rounded up and is counted as one second.

NOTE
If the life is counted every 0.1 second (bit 0 (FCO) of parameter No. 6805 is set to 1), cutting time less than 0.1 second is always rounded up and is counted as 0.1 second.

#7 TAD With tool change type D (bit 7 (M6E) of parameter No. 6801 is set to 1), when a block specifying M06 contains no T command:
0: An alarm PS0153, "T-CODE NOT FOUND" is issued.
1: No alarm is issued.

6810	Tool life management ignore number

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 99999999
This parameter sets the tool life management ignore number.
When the value specified in a T code exceeds the value set in this parameter, the value obtained by subtracting the parameter-set value from the T code value is assumed to be the tool group number for tool life management.

6811	Tool life count restart M code

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 127 (except 01, 02, 30, 98, and 99)
When 0 is specified, it is ignored.
When the life is specified by count, the tool change signal TLCH <Fn064.0> is output if the life of at least one tool group has expired when the tool life count restart M code is issued.
The T code (tool life management group command) specified after the tool life count restart M code selects a tool whose life has not expired from a specified group, and the next M06 command increments the tool life counter by one.
When the life is specified by duration, specifying the tool life count restart M code causes nothing. When 0 is set in this parameter, the tool life count restart M code is invalid. When the data of M code exceeds 127 values, set 0 in parameter No. 6811, and set the value of M code in parameter No. 13221. The data range of parameter No. 13221 is from 0 to 255.

NOTE
The use of this parameter varies depending on whether the tool management function or tool life management function is used.

6813	Maximum number of groups in tool life management

NOTE
After this parameter has been set, the power must be turned off then back on for the setting to become effective.

[Input type] Parameter input
[Data type] Word path
[Unit of data] Group
[Valid data range] 0, 8, 16 to 256,1024 (when the additional tool life management group option is added)

This parameter sets the maximum number of groups to be used for each path. As the maximum number of groups, set a multiple of eight. When the tool life management function is not used, 0 must be set. Set this parameter so that the total number of groups in all paths does not exceed the total number of groups in the entire system (256 groups). When the additional tool life management group option is added, set this parameter so that the total number of groups in all paths does not exceed 1024.

NOTE
When the power is turned on, all tool life management file data is initialized. So, tool life management data must be set for all paths that use tool life management.

6844	Remaining tool life (use count)

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 9999
This parameter sets a remaining tool life (use count) used to output the tool life notice signal TLCHB <Fn064.3> when the tool life is specified by use count. If a value greater than the tool life value or 0 is set in this parameter, the tool life notice signal is not output.

6845	Remaining tool life (use duration)

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min
[Valid data range] Not greater than the tool life value
This parameter sets the remaining tool life (use duration) used to output the tool life notice signal TLCHB <Fn064.3> when the tool life is specified by use duration. If a value greater than the tool life value or 0 is specified in this parameter, the tool life notice signal is not output.

NOTE
When the life is counted every 0.1 second (bit 0 (FCO) of parameter No. 6805 = 1), the parameter value is in 0.1-minute increments.

6846	
	Remaining tool number in a group

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 127
This parameter sets the remaining tool number in a group.
When the remaining tool number in the group selected by the T code is smaller than or equal to the value set in this parameter, the remaining tool number notice signal TLAL <Fn154.0> is output. When this parameter is set to 0, the remaining tool number notice signal is not output.

6930	Maximum value of the operating range of the 1st position switch (PSW101)
6931	Maximum value of the operating range of the 2nd position switch (PSW102)
to	to
6945	Maximum value of the operating range of the 16th position switch (PSW116)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the maximum value of the operating range of the first to sixteenth position switches.

NOTE
1 For a diameter-specified axis, use radius values to specify the parameters used to set the maximum and minimum values of an operating range.
2 The position switch function is enabled upon completion of reference position return.

6950	Minimum value of the operating range of the 1st position switch (PSW201)
6951	Minimum value of the operating range of the 2nd position switch (PSW202)
to	to
6965	Minimum value of the operating range of the 16th position switch (PSW216)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set the minimum value of the operating range of the first to sixteenth position switches.

NOTE
1 For a diameter-specified axis, use radius values to specify the parameters used to set the maximum and minimum values of an operating range.
2 The position switch function is enabled upon completion of reference position return.

	#7	#6	#5	#4	#3	#2	#1	#0
7001						JST		MIT

[Input type] Parameter input
[Data type] Bit path

 # 0 MIT Manual intervention and return function is:
0: Disabled.
1: Enabled.

#2 JST In manual numerical specification, the cycle start lamp signal STL <Fn000.5> is:
0: Not output.
1: Output.

	#7	#6	#5	#4	#3	#2	#1	#0
7002	TRO	TNR			JBF	JTF	JSF	JMF

[Input type] Parameter input
[Data type] Bit path

#0 JMF In manual numerical specification, M function specification is:
0: Allowed.
1: Not allowed.

#1 JSF In manual numerical specification, S function specification is:
0: Allowed.
1: Not allowed.

#2 JTF In manual numerical specification, T function specification is:
0: Allowed.
1: Not allowed.

#3 JBF In manual numerical specification, B function specification is:
0: Allowed.
1: Not allowed.

#6 TNR When the updated compensation value in the tool retract and recover function is effective (the bit 7 (TRO) of parameter No.7002 is set to 1)
0: The updated compensation value is effective in the recovery operation.
1: The updated compensation value is effective in the re-positioning operation.

#7 TRO When the compensation value is updated while the tool retract and recover function is executing,
0: The updated compensation value is invalid.
1: The updated compensation value is effective.

	#7	#6	#5	#4	#3	#2	#1	#0
7040						RPS		

[Input type] Parameter input
[Data type] Bit path

#2 RPS When the tool retract signal TRESC <Gn059.0> is set to “1” after G10.6 is specified alone:

0: The tool is not retracted.
1: The tool is retracted with the value set for parameter No. 7041 or 11261 used as the incremental retraction distance.

7041	Retraction distance in tool retract and return

[Input type] Setting input
[Data type] Real axis
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the retraction distance used when G10.6 is specified alone for the tool retract and return. The tool is retracted by the distance set for this parameter in the incremental mode. This data is valid only when bit 2 (RPS) of parameter No. 7040 is set to 1.
During tool center point control and workpiece setting error compensation, however, this parameter is valid if parameter No. 11261 is set to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
7055					BCG			

[Input type] Parameter input
[Data type] Bit path

#3 BCG The pre-interpolation bell-shaped acceleration/deceleration time constant change function is:

0: Disabled.
1: Enabled.

7066	Acceleration/deceleration reference speed for the time constant change function of bell-shaped acceleration/deceleration before interpolation

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter is used when the time constant change function of bell-shaped acceleration/deceleration before interpolation is used.

	#7	#6	#5	#4	#3	#2	#1	#0
7100					HCL		THD	JHD

[Input type] Parameter input
[Data type] Bit path

#0 JHD Manual handle feed in JOG feed mode or incremental feed in the manual handle feed is:

0: Invalid.
1: Valid.

#1 THD In the TEACH IN JOG mode, the manual pulse generator is:
0: Disabled.
1: Enabled.

#3 HCL The clearing of handle interruption amount display by soft key [CAN] operation is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
7102								HNGx

[Input type] Parameter input
[Data type] Bit axis

#0 HNGx Axis movement direction for rotation direction of manual pulse generator
0: Same in direction
1: Reverse in direction

	#7	#6	#5	#4	#3	#2	#1	#0
7103					HIT	HNT	RTH	

[Input type] Parameter input
[Data type] Bit path

#1 RTH By a reset or emergency stop, the amount of manual handle interruption is:
0: Not canceled.
1: Canceled.

#2 HNT When compared with the travel distance magnification selected by the manual handle feed travel distance selection signals (incremental feed signals) (MP1, MP2 <Gn019.4, Gn019.5>), the travel distance magnification for incremental feed/manual handle feed is:
0: Same.
1: 10 times greater.

#3 HIT When compared with the travel distance magnification selected by the manual handle feed travel distance selection signals (incremental feed signals (MP1, MP2 <Gn019.4, Gn019.5>), the travel distance magnification for manual handle interrupt is:
0: Same.
1: 10 times greater.

7117	Allowable number of pulses that can be accumulated during manual handle feed

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Pulse
[Valid data range] 0 to 999999999
This parameter sets the number of pulses from the manual pulse generator that exceed the rapid traverse rate and can be accumulated without being discarded if manual handle feed faster than the rapid traverse rate is specified.
0:
The feedrate is clamped to the rapid traverse rate. Those handle pulses that exceed the rapid traverse rate are ignored. (The scale reading of the manual pulse generator may not match the travel distance.)

Other than 0:

The feedrate is clamped to the rapid traverse rate. However, those handle pulses that exceed the rapid traverse rate are not ignored. In connection with the manual handle feed travel distance selection signals MP1 and MP2 <Gn019.4, Gn019.5>, the incremental feed amount is determined as described below. (Even if the rotation of the manual pulse generator is stopped, the tool stops after moving by the number of pulses accumulated in the CNC.)

Magnification set by manual handle feed amount selection signals MP1, MP2 <Gn019.4, Gn019.5> is m, value of parameter No.7117 is n.

n < m: Clamping is set performed at value of parameter No.7117.

n ≥ m: Amount A+B, showed in figure, which's value is multiple of m and small than n. As a result, clamping is performed as an integral multiple of the selected magnification.

7160	Approach handle clamp feedrate

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis。
[Valid data range] Refer to standard parameter setting table (C)
Approach handle clamp feedrate is set.

7161	Guidance handle clamp feedrate

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis。
[Valid data range] Refer to standard parameter setting table (C)
Guidance handle clamp feedrate is set.

	#7	#6	#5	#4	#3	#2	#1	#0
7200		OP7	OP6	OP5	OP4	OP3	OP2	OP1

[Input type] Parameter input
[Data type] Bit path

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 OP1 Mode selection on software operator's panel
0: Not performed
1: Performed

#1 OP2 JOG feed axis select and manual rapid traverse select on software operator's panel
0: Not performed
1: Performed

#2 OP3 Manual pulse generator's axis select and manual pulse generator's magnification select on software operator's panel
0: Not performed
1: Performed

#3 OP4 JOG feedrate override select, feedrate override select, and rapid traverse override select on software operator's panel
0: Not performed
1: Performed

#4 OP5 Optional block skip select, single block select, machine lock select, and dry run select on software operator's panel
0: Not performed
1: Performed

#5 OP6 Protect key on software operator's panel
0: Not performed
1: Performed

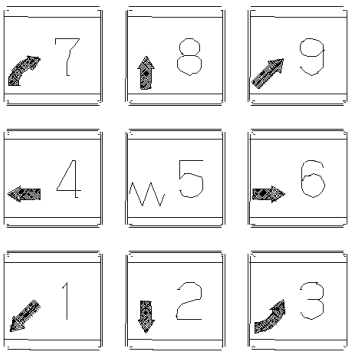
#6 OP7 Feed hold on software operator's panel
0: Not performed
1: Performed

7210	**Jog-movement axis and its direction on software operator's panel "↑"**
7211	**Jog-movement axis and its direction on software operator's panel "↓"**
7212	**Jog-movement axis and its direction on software operator's panel "→"**
7213	**Jog-movement axis and its direction on software operator's panel "←"**
7214	**Jog-movement axis and its direction on software operator's panel "↙"**
7215	**Jog-movement axis and its direction on software operator's panel "↗"**
7216	**Jog-movement axis and its direction on software operator's panel "⤴"**
7217	**Jog-movement axis and its direction on software operator's panel "⤴"**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 8
On software operator's panel, set a feed axis corresponding to an arrow key on the MDI unit when jog feed is performed.

Setting value	Feed axis and direction
0	Not moved
1	First axis, positive direction
2	First axis, negative direction
3	Second axis, positive direction
4	Second axis, negative direction
5	Third axis, positive direction
6	Third axis, negative direction
7	Fourth axis, positive direction
8	Fourth axis, negative direction

Arrow keys on MDI unit



[Example] Under X, Y, and Z axis configuration, to set arrow keys to feed the axes in the direction specified as follows, set the parameters to the values given below. <8↑> to the positive direction of the Z axis, <2↓> to the negative direction of the Z axis, <6→> to the positive direction of the X axis <4←> to the negative direction of the X axis, <1↙> to the positive direction of the Y axis, <9↗> to the negative direction of the Y axis
Parameter No.7210 = 5 (Z axis, positive direction)
Parameter No.7211 = 6 (Z axis, negative direction)
Parameter No.7212 = 1 (X axis, positive direction)
Parameter No.7213 = 2 (X axis, negative direction)
Parameter No.7214 = 3 (Y axis, positive direction)
Parameter No.7215 = 4 (Y axis, negative direction)
Parameter No.7216 = 0 (Not used)
Parameter No.7217 = 0 (Not used)

	#7	#6	#5	#4	#3	#2	#1	#0
7300	MOU	MOA	CCS					

[Input type] Parameter input
[Data type] Bit path

#5 CCS When the Cs contour control axis is used in the spindle mode or when the origin of the Cs contour control axis is not established, program restart is:
0: Disabled.
1: Enabled.

#6 MOA In program restart operation, before movement to a machining restart point:
0: The last M, S, T, and B codes are output.
1: All M codes and the last S, T, and B codes are output.
This parameter is enabled when the bit 7 (MOU) of parameter No.7300 is set to 1.

#7 MOU In program restart operation, before movement to a machining restart point after restart block search:
0: The M, S, T, and B codes are not output.
1: The last M, S, T, and B codes are output.

	#7	#6	#5	#4	#3	#2	#1	#0
7301							3DD	

[Input type] Parameter input
[Data type] Bit path

#1 3DD In program restart operation, when the restart block is in 3-dimensional coordinate conversion mode G68 (machining center system) or G68.1 (lathe system), the tool moves to the restart point along each axis:
0: According to the program coordinate system in dry run.
1: According to the workpiece coordinate system in dry run.
The restart coordinates and restart travel distance are also displayed in the coordinate system set in this parameter.

NOTE
The change made to this parameter in program restart operation is ignored.

7310	Ordinal number of an axis along which a movement is made in dry run after program restart

[Input type] Setting input
[Data type] Byte axis
[Valid data range] 1 to (Number of controlled axes)
This parameter sets the ordinal number of an axis along which a movement is made in dry run after the program is restarted.

	#7	#6	#5	#4	#3	#2	#1	#0
7501	IPC	IT2	IT1	IT0	BDS			

[Input type] Parameter input
[Data type] Bit path

#3 BDS In the high-speed binary program operation mode, the data format of axis movement is:
0 : Special format.
1 : Ordinary format.

- When BDS is set to 0 (special format)
  The bits marked with an asterisk (*) are used to specify the travel distance per unit time.

15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
*	*	*	*	*	*	*	0	*	*	*	*	*	*	*	0

- When BDS is set to 1 (ordinary format)
  The bits marked with an asterisk (*) are used to specify the travel distance per unit time.

15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
*	*	*	*	*	*	*	*	*	*	*	*	*	*	*	*

#4 IT0
#5 IT1
#6 IT2

IT2	IT1	IT0	Interpolation of high-speed cutting G05 data (ms)
0	0	0	8
0	0	1	2
0	1	0	4
0	1	1	1
1	0	0	16
1	1	1	0.5

NOTE
To perform high-speed cycle cutting for multiple paths, set the same interpolation for all paths.

#7 IPC While high-speed cutting (G05) is being executed using cycle machining data, monitoring to see whether distribution processing is stopped or not is:
0: Not performed.
1: Performed. (If distribution processing is stopped, an alarm PS0179, "PARAM. (NO.7510) SETTING ERROR" is issued after distribution is completed.)

	#7	#6	#5	#4	#3	#2	#1	#0
7502			LC2	LC1				

[Input type] Parameter input

[Data type] Bit path

#4 LC1
#5 LC2

LC2	LC1	End timing of servo learning function during high-speed cycle cutting retract function
0	0	Disables the servo learning function, after which retract operation starts.
0	1	Disables the servo learning function upon the completion of retract operation.
1	0	Disables the servo learning function upon the completion of a retract cycle.

	#7	#6	#5	#4	#3	#2	#1	#0
7503				HCT				

[Input type] Parameter input
[Data type] Bit path

#4 HCT The variable number for starting storage of high-speed cycle machining distribution data is:
0: Specified by 1/10.
1: Specified by 1/100.

	#7	#6	#5	#4	#3	#2	#1	#0
7504								BM0

[Input type] Parameter input
[Data type] Bit path

#0 BM0 During high-speed cycle cutting or high-speed binary program operation, axis moving signals MV1 to MV8 <Fn102> are:
0: Always set to "1".
1: Set to "1" when the tool moves along the axis.

NOTE
When bit 0 (BM0) of parameter No. 7504 is set to 1 and the same position is specified after a very small movement for each execution cycle, axis moving signals MV1 to MV8 <Fn102> may not be output.

	#7	#6	#5	#4	#3	#2	#1	#0
7505							HUNx	

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter bit is set, the power must be turned off before operation is continued.

#1 HUNx During high-speed cutting, the setting unit of distribution data is:
0: 1 pulse.
1: 10 pulses.

NOTE
If data distributed at one time is longer than one word because of the least input increment and the maximum feedrate, this parameter is used. If bit parameter HUN is set to 1 for an axis, high-speed cycle machining/high-speed binary program operation data distributed to that axis is internally multiplied by 10 by the CNC and then input. Therefore, when using HUN for an axis, set a 1/10 of high-speed cycle machining/high-speed binary program operation data to be distributed to that axis.

7510 **Number of controlled axes for high-speed cycle machining or high-speed binary program operation**

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to number of controlled axes
Set the number of axes to be controlled when G05 is specified to perform high-speed cycle machining or high-speed binary program operation.

7514 **Retract direction and retract feedrate in high-speed cycle machining retract operation**

[Input type] Parameter input
[Data type] Real axis
[Valid data range] Refer to standard parameter setting table (C)
This parameter sets a retract direction and retract feedrate along each axis in a high-speed cycle machining retract operation. The superimposed feedrate for each axis is specified, and the retract direction is specified with a sign.

7515 **Number of retract operation distributions in a high-speed cycle machining retract operation**

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 99999999
This parameter sets the number of retract operation distributions in a high-speed cycle machining retract operation.
When the cycle currently being executed ends before the number of distributions specified in this parameter are performed, retract operation is terminated. When 0 is specified in this parameter, the number of retract operation distributions is assumed to be infinite. In this case, retract operation is performed until the cycle currently being executed ends.

7516 **High-speed cycle cutting data area ID**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
When data assigned to a path is to be used as common data, this parameter sets the number of the path.

7517 **Number of high-speed cycle cutting data items**

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 65536, 0 to 131072, 0 to 262144, 0 to 786432, 0 to 2000000
This parameter sets the number of data items to be used for high-speed cycle cutting.

Since variables are sequentially assigned to each path from path 1 to path 2 and so on, to some paths, the specified number of variables may not be assigned depending on the setting. When this parameter is set to 0 for all paths, however, all variables are assigned to path 1.
In parameter No. 7516, set the number of the path of which data is to be used as common data. For the path of which data is to be used as common data, set the parameter to 0.

[Example 1] When the high-speed cycle cutting data variable mode is set to standard (65536 variables), 40000 variables are to be used for paths 1 and 3 as common data and remaining 25536 variables are to be used for path 2

	No.7516	No.7517	Available variables
Path 1	0	40000	#20000 to #59999
Path 2	0	25536	#20000 to #45535
Path 3	1	0	#20000 to #59999

[Example 2] When the high-speed cycle cutting data variable mode is set to addition B (262144 variables) and high-speed cycle cutting is to be performed only for path 2

	No.7516	No.7517	Available variables
Path 1	0	0	#200000 to #462143 (Not used)
Path 2	1	0	#200000 to #462143
Path 3	0	0	-

or

	No.7516	No.7517	Available variables
Path 1	0	0	-
Path 2	0	262144	#200000 to #462143
Path 3	0	0	-

[Example 3] When the high-speed cycle cutting data variable mode is addition D (2000000 variables) and all variables are to be used as common variables

	No.7516	No.7517	Available variables
Path 1	0	0	#2000000 to #3999999
Path 2	1	0	#2000000 to #3999999
Path 3	1	0	#2000000 to #3999999

	#7	#6	#5	#4	#3	#2	#1	#0
7600	PLZ							

[Input type] Parameter input
[Data type] Bit path

#7 PLZ Reference position return based on a G28 command on the tool rotation axis for polygon turning is:
0: Performed in the same sequence as manual reference position return.
1: Performed by positioning using the rapid traverse rate.
The synchronous axis returns to the reference position in the same sequence as the manual reference position return when no return-to-reference position is performed after the power is turned on.

7610	Control axis number of tool rotation axis for polygon turning

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path

[Valid data range] 1 to number of controlled axes
This parameter sets the control axis number of a rotation tool axis used for polygon turning.
However, when a G51.2 command is executed by setting 0 in this parameter, operation stops with the alarm PS0314, “ILLEGAL SETTING OF POLYGONAL AXIS”.

7640	Master axis in spindle-spindle polygon turning

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Maximum number of controlled axes (Within a path)
This parameter sets the master axis in spindle-spindle polygon turning.

NOTE
1 Spindle-spindle polygon turning is enabled only for serial spindles.
2 When any one of parameters No. 7640 and 7641 is set to 0, polygon turning is performed using the first spindle (master axis) and the second spindle (polygon synchronous axis) in the path to which the parameter belongs.
3 When a spindle other than the first serial spindle is used as a master axis, the multi-spindle control option is required to specify an S command for the master axis.
4 When the PMC window function or G10 command is used to rewrite this parameter, rewrite this parameter before the block specifying the spindle-spindle polygon command G51.2. When the PMC window function is used to rewrite this parameter in the block immediately before G51.2, specify the rewriting of this parameter by using an M code (parameter No. 3411 and up) without buffering.

7641	Polygon synchronous axis in spindle-spindle polygon turning

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Maximum number of controlled axes (Within a path)
This parameter sets the polygon synchronous (slave) axis in spindle-spindle polygon turning.

NOTE
1 Spindle-spindle polygon turning is enabled only for serial spindles.
2 When any one of parameter No. 7640 and No. 7641 is set to 0, polygon turning is performed using the first spindle (master axis) and the second spindle (polygon synchronous axis) in the path to which the parameter belongs.
3 When a spindle other than the first serial spindle is used as a master axis, the multi-spindle control option is required to specify an S command for the master axis.
4 When the PMC window function or G10 command is used to rewrite this parameter, rewrite this parameter before the block specifying the spindle-spindle polygon command G51.2. When the PMC window function is used to rewrite this parameter in the block immediately before G51.2, specify the rewriting of this parameter by using an M code (parameter No. 3411 and up) without buffering.

7642	Master axis in spindle-spindle polygon turning (spindle number common to the system)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Maximum number of controlled axes (Common to the system)
This parameter sets the master axis in spindle-spindle polygon turning.

NOTE
1 Spindle-spindle polygon turning is enabled only for serial spindles.
2 This parameter is invalid if either parameter No. 7642 or No.7643 is set to 0. In this case, the settings of parameter No. 7640 and No.7641 are valid.
3 When a spindle other than the first serial spindle is used as a master axis, the multi-spindle control option is required to specify an S command for the master axis.
4 When the PMC window function or G10 command is used to rewrite this parameter, rewrite this parameter before the block specifying the spindle-spindle polygon command G51.2. When the PMC window function is used to rewrite this parameter in the block immediately before G51.2, specify the rewriting of this parameter by using an M code (parameter No. 3411 and up) without buffering.
5 A spindle number common to the system is to be set in this parameter. When using this parameter, set 0 in parameter No. 7640 and No. 7641.

7643	Polygon synchronous axis in spindle-spindle polygon turning (spindle number common to the system)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Maximum number of controlled axes (Common to the system)
This parameter sets the polygon synchronous (slave) axis in spindle-spindle polygon turning.

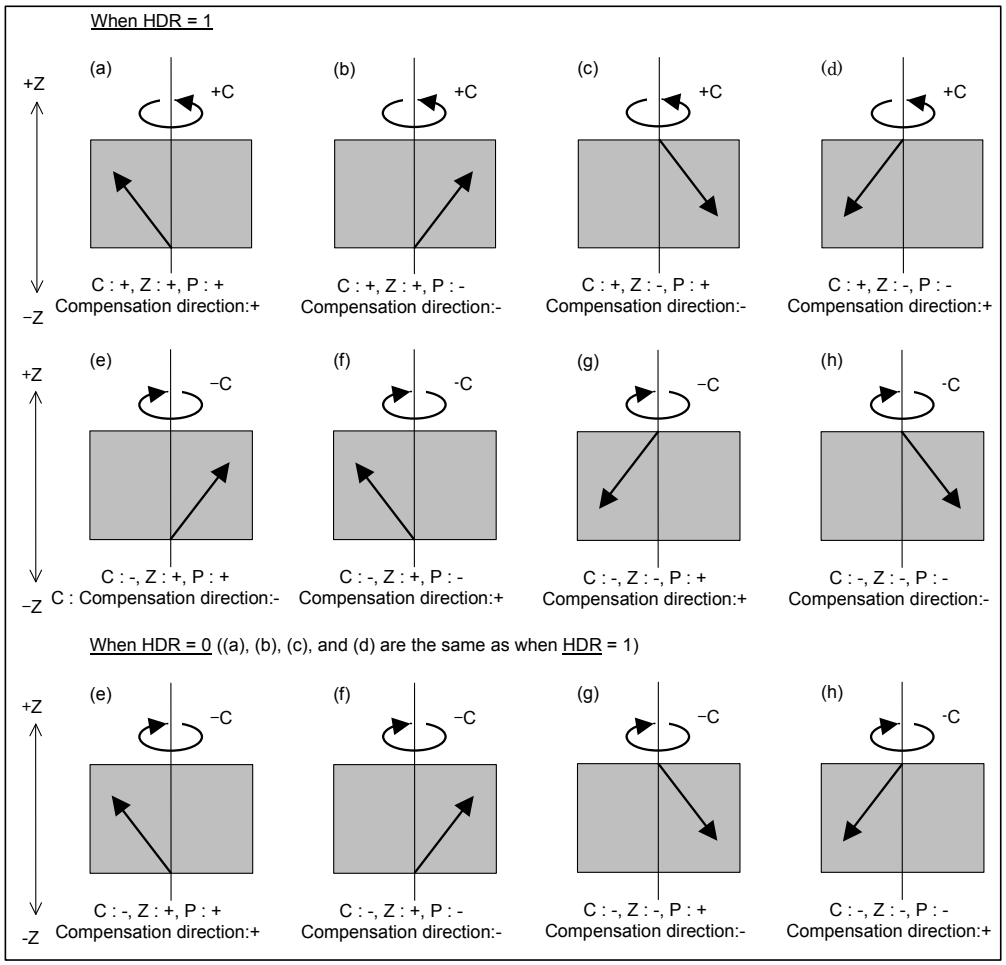
NOTE
1 Spindle-spindle polygon turning is enabled only for serial spindles.
2 This parameter is invalid if either parameter No. 7642 or No.7643 is set to 0. In this case, the settings of parameter No. 7640 and No.7641 are valid.
3 When a spindle other than the first serial spindle is used as a master axis, the multi-spindle control option is required to specify an S command for the master axis.
4 When the PMC window function or G10 command is used to rewrite this parameter, rewrite this parameter before the block specifying the spindle-spindle polygon command G51.2. When the PMC window function is used to rewrite this parameter in the block immediately before G51.2, specify the rewriting of this parameter by using an M code (parameter No. 3411 and up) without buffering.
5 A spindle number common to the system is to be set in this parameter. When using this parameter, set 0 in parameter No. 7640 and No. 7641.

	#7	#6	#5	#4	#3	#2	#1	#0
7700						HDR		HBR

[Input type] Parameter input
[Data type] Bit path

#0 HBR When the electronic gear box (EGB) function is used, performing a reset:
0: Cancels the synchronization mode (G81 or G81.5).
1: Does not cancel the synchronization mode. The mode is canceled only by the G80 or G80.5 command.

#2 HDR Direction of helical gear compensation (usually, set 1.)
[Example]
To cut a left-twisted helical gear when the direction of rotation about the C-axis is the negative (-) direction:
0: Set a negative (-) value in P.
1: Set a positive (+) value in P.



	#7	#6	#5	#4	#3	#2	#1	#0
7701					LZR			

[Input type] Parameter input
[Data type] Bit path

#3 LZR When L (number of hob threads) = 0 is specified at the start of EGB synchronization (G81):

0: Synchronization is started, assuming that L = 1 is specified.
1: Synchronization is not started, assuming that L = 0 is specified. However, helical gear compensation is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
7702	PHD	PHS			ART			TDP

[Input type] Parameter input
[Data type] Bit path

#0 TDP The specifiable number of teeth, T, of the electronic gear box (G81) is:

0: 1 to 1000
1: 0.1 to 100 (1/10 of a specified value)

NOTE
In either case, a value from 1 to 1000 can be specified.

#3 ART The retract function executed when an alarm is issued is:

0: Disabled.
1: Enabled.

When an alarm is issued, a retract operation is performed with a set feedrate and travel distance (parameters Nos. 7740 and 7741).

NOTE
If a servo alarm is issued for other than the axis along which a retract operation is performed, the servo activating current is maintained until the retract operation is completed.

#6 PHS When the G81/G80 block contains no R command:

0: Acceleration/deceleration is not performed at the start or cancellation of EGB synchronization.
1: Acceleration/deceleration is performed at the start or cancellation of EGB synchronization. After acceleration at the start of synchronization, phase synchronization is automatically performed.

#7 PHD The direction of movement for automatic phase synchronization is:

0: Positive (+).
1: Negative (-).

	#7	#6	#5	#4	#3	#2	#1	#0
7703						ARO	ARE	ERV

[Input type] Parameter input
[Data type] Bit path

#0 ERV During EGB synchronization (G81), feed per revolution is performed for:

0: Feedback pulses.
1: Pulses converted to the speed for the workpiece axis.

#1 ARE The retract function executed when an alarm is issued retracts the tool during:

0: EGB synchronization or automatic operation (automatic operation signal OP <Fn000.7> = "1").
1: EGB synchronization.

#2 ARO The retract function executed when an alarm is issued retracts the tool during:

0: EGB synchronization.
1: EGB synchronization and automatic operation (automatic operation signal OP = "1").

The following table lists the parameter settings and corresponding operation.

ARE	ARO	Operation
1	0	During EGB synchronization
1	1	During EGB synchronization and automatic operation
0	0	During EGB synchronization or automatic operation
0	1	

NOTE
1 Parameters ARE and ARO are valid when bit 3 (ART) of parameter No. 7702 is set to 1 (when the retract function executed when an alarm is issued).
2 This parameter is valid when bit 1 (ARE) of parameter No. 7703 is set to 1.

7710	Axis number of an axis to be synchronized using the method of command specification for a hobbing machine

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to Number of controlled axes

When there are several groups of axes to be synchronized (the axes for which bit 0 (SYNMOD) of parameter No. 2011 is set to 1), an axis for which to start synchronization is specified using the following command (for a hobbing machine):

G81 T t L ± *l* ;

t: Spindle speed ($1 \leq t \leq 5000$)

l: Number of synchronized axis rotations ($-250 \leq l \leq 250$)

Synchronization between the spindle and a specified axis is established with the ratio of ±*l* rotations about the synchronized axis to t spindle rotations.

t and *l* correspond to the number of teeth and the number of threads on the hobbing machine, respectively.

Above command is issued without setting this parameter when there are several groups of axes to be synchronized, the alarm PS1593, “EGB PARAMETER SETTING ERROR” is issued.

NOTE
The setting of this parameter becomes valid after the power is turned off then back on.

	#7	#6	#5	#4	#3	#2	#1	#0
7731		EPA			ECN			EFX

[Input type] Parameter input
[Data type] Bit path

#0 EFX As the EGB command:
0: G80 and G81 are used.
1: G80.4 and G81.4 are used.

NOTE
When this parameter is set to 0, no canned cycle for drilling can be used.

#3 ECN When the automatic phase synchronization function for the electronic gear box is disabled, during EGB synchronization, the G81 or G81.5 command:
0: Cannot be issued again. (The alarm PS1595, “ILL-COMMAND IN EGB MODE” is issued.)
1: Can be issued again.

#6 EPA Automatic phase synchronization for the electronic gear box is performed in such a way that:

0: The machine coordinate 0 of the slave axis is aligned to the position of the master axis one-rotation signal.

1: The position of the slave axis at synchronization start is aligned to the position of the master axis one-rotation signal. (Specification of Series 16*i*)

7740	Feedrate during retraction

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the feedrate during retraction for each axis.

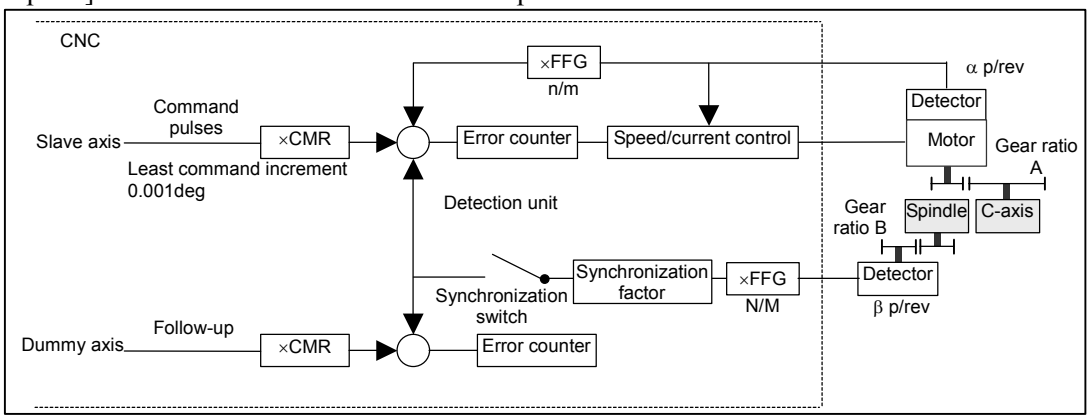
7772	Number of position detector pulses per rotation about the tool axis

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Detection unit
[Valid data range] 1 to 999999999
This parameter sets the number of pulses per rotation about the tool axis (on the spindle side), for the position detector.
For an A/B phase detector, set this parameter with four pulses equaling one A/B phase cycle.

7773	Number of position detector pulses per rotation about the workpiece axis

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Detection unit
[Valid data range] 1 to 999999999
This parameter sets the number of pulses per rotation about the workpiece axis (on the slave side), for the position detector.
Set the number of pulses output by the detection unit.
Set parameters Nos. 7772 and 7773 when using the G81 EGB synchronization command.

[Example 1] When the EGB master axis is the spindle and the EGB slave axis is the C-axis



Gear ratio of the spindle to the detector B:
1/1 (The spindle and detector are directly connected to each other.)
Number of detector pulses per spindle rotation β: 80,000 pulses/rev
(Calculated for four pulses for one A/B phase cycle)
FFG N/M of the EGB dummy axis: 1/1
Gear ratio of the C-axis A: 1/36 (One rotation about the C-axis to 36 motor rotations)
Number of detector pulses per C-axis rotation α: 1,000,000 pulses/rev
C-axis CMR: 1
C-axis FFG n/m: 1/100

In this case, the number of pulses per spindle rotation is:
$80000 \times 1/1 = 80000$
Therefore, set 80000 for parameter No. 7772.
The number of pulses per C-axis rotation in the detection unit is:
$1000000 \div 1/36 \times 1/100 = 360000$
Therefore, set 360000 for parameter No. 7773.

[Example 2] When the gear ratio of the spindle to the detector B is 2/3 for the above example (When the detector rotates twice for three spindle rotations)
In this case, the number of pulses per spindle rotation is:

$$80000 \times \frac{2}{3} = \frac{160000}{3}$$

160000 cannot be divided by 3 without a remainder. In this case, change the setting of parameter No. 7773 so that the ratio of the settings of parameters Nos. 7772 and 7773 indicates the value you want to set.

$$\frac{\text{No.7772}}{\text{No.7773}} = \frac{160000 \diagup 3}{360000} = \frac{160000}{360000 \times 3} = \frac{160000}{1080000}$$

Therefore, set 160000 for parameter No. 7772 and 1080000 for parameter No. 7773.
As described above, all the settings of parameters Nos. 7772 and 7773 have to do is to indicate the ratio correctly. So, you can reduce the fraction indicated by the settings. For example, you may set 16 for parameter No. 7772 and 108 for parameter No. 7773 for this case.

7776	Feedrate during automatic phase synchronization for the workpiece axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg/min
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the feedrate during automatic phase synchronization for the workpiece axis.
When this parameter is set to 0, the rapid traverse rate (parameter No. 1420) is used as the feedrate during automatic phase synchronization.

7777	Angle shifted from the spindle position (one-rotation signal position) which the workpiece axis uses as the reference of phase synchronization

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0.000 to 360.000 (when the increment system is IS-B)

This parameter sets the angle shifted from the spindle position (one-rotation signal position) which the workpiece axis uses as the reference of phase synchronization.

7778	Acceleration for acceleration/deceleration for the workpiece axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg/sec^2
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(For a millimeter machine, 0.0 to +100000.0, for an inch machine, 0.0 to +10000.0)
This parameter sets an acceleration for acceleration/deceleration for the workpiece axis.

NOTE
1 In the Series 16*i*, acceleration/deceleration for automatic phase matching is set by specifying a feedrate and a time constant in parameters Nos. 2135 and 2136 (Nos. 4384 and 4385 in the case of spindle EGB) separately; in the Series 30*i*, acceleration/deceleration is directly set in parameter No. 7778.
2 If this parameter is set to 0, specifying G81 results in an alarm PS1598, “EGB AUTO PHASE PARAMETER SETTING ERROR”.

7782	Number of pulses from the position detector per EGB master axis rotation

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 999999999
For a slave axis, set the number of pulses generated from the position detector per EGB master axis rotation.
For an A/B phase detector, set this parameter with four pulses equaling one A/B phase cycle.

7783	Number of pulses from the position detector per EGB slave axis rotation

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 999999999

For a slave axis, set the number of pulses generated from the position detector per EGB slave axis rotation.
Set the number of pulses output by the detection unit.
Set this parameter when using the G81.5 EGB synchronization command.
The method for setting parameters Nos. 7782 and 7783 is the same as for parameters Nos. 7772 and 7773. For the method, see the description of parameters Nos. 7772 and 7773.

The ratio of the number of pulses for the master slave to that of pulses for the slave axis may be valid, but the settings of the parameters may not indicate the actual number of pulses. For example, the number of pulses may not be able to be divided without a remainder for the reason of the master and slave axis gear ratios as described in example 2. In this case, the following methods cannot be used for the G81.5 command:

G81.5 T_ C_ ; When the speed is specified for the master axis and the travel distance is specified for the slave axis
G81.5 P_ C0 L_ ; When the number of pulses is specified for the master axis and the speed is specified for the slave axis

	#7	#6	#5	#4	#3	#2	#1	#0
8001					RDE	OVE		MLE

[Input type] Parameter input
[Data type] Bit path

#0 MLE Whether all axis machine lock signal MLK <Gn108> is valid for PMC-controlled axes
0: Valid
1: Invalid
The axis-by-axis machine lock signal MLKx depends on the setting of bit 1 of parameter No. 8006.

#2 OVE Signals related to dry run and override used in PMC axis control
0: Same signals as those used for the CNC
1: Signals specific to the PMC
The signals used depend on the settings of these parameter bits as indicated below.

Signals	Bit 2 (OVE) of parameter No. 8001 = 0 (same signals as those used for the CNC)		Bit 2 (OVE) of parameter No. 8001 = 1 (signals specific to the PMC)	
Feedrate override signals	*FV0 to *FV7	<G012>	*EFOV0 to *EFOV7	<G151>
Override cancellation signal	OVC	<G006.4>	EOVC	<G150.5>
Rapid traverse override signals	ROV1,2	<G014.0,1>	EROV1,2 or *EROV0 to *EROV7	<G150.0,1> <G151>
Dry run signal	DRN	<G046.7>	EDRN	<G150.7>
Rapid traverse selection signal	RT	<G019.7>	ERT	<G150.6>

(The listed signal addresses when PMC signals are selected are for the 1st group. Actual addresses differs depending on the used group.)

#3 RDE Whether dry run is valid for rapid traverse in PMC axis control
0: Invalid
1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
8002	FR2	FR1	PF2	PF1	F10			RPD

[Input type] Parameter input
[Data type] Bit path

#0 RPD Rapid traverse rate for PMC-controlled axes
0: Feedrate specified with parameter No. 1420
1: Feedrate specified with the feedrate data in an axis control command by PMC

#3 F10 Least increment for the feedrate for cutting feed (per minute) in PMC axis control
The following settings are applied when bit 4 (PF1) of parameter No. 8002 is set to 0 and bit 5 (PF2) of parameter No. 8002 is set to 0.

	F10	IS-A	IS-B	IS-C	IS-D	IS-E
Millimeter input (mm/min)	0	10	1	0.1	0.01	0.001
	1	100	10	1	0.1	0.01
Inch input (inch/min)	0	0.1	0.01	0.001	0.0001	0.00001
	1	1	0.1	0.01	0.001	0.0001

#4 PF1

#5 PF2 Set the feedrate unit of cutting feedrate (feed per minute) for an axis controlled by the PMC.

Bit 5 (PF2) of parameter No. 8002	Bit 4 (PF1) of parameter No. 8002	Feedrate unit
0	0	1 / 1
0	1	1 / 10
1	0	1 / 100
1	1	1 / 1000

#6 FR1

#7 FR2 Set the feedrate unit for cutting feedrate (feed per rotation) for an axis controlled by the PMC.

Bit 7 (FR2) of parameter No. 8002	Bit 6 (FR1) of parameter No. 8002	Millimeter input (mm/rev)	Inch input (inch/rev)
0	0	0.0001	0.000001
1	1		
0	1	0.001	0.00001
1	0	0.01	0.0001

	#7	#6	#5	#4	#3	#2	#1	#0
8004		NCI						

[Input type] Parameter input
[Data type] Bit path

#6 NCI In axis control by the PMC, a position check at the time of deceleration is:
0: Performed.
1: Not performed.

	#7	#6	#5	#4	#3	#2	#1	#0
8005								EDC

[Input type] Setting input
[Data type] Bit path

#0 EDC In axis control by the PMC, an external deceleration function is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
8006		EZR		EFD			MLS	

[Input type] Parameter input
[Data type] Bit path

#1 MLS When bit 0 (MLE) of parameter No. 8001 is set to 1 (to disable the all axis machine lock signal) in PMC axis control, axis-by-axis machine lock is:
0: Disabled.
1: Enabled.

#4 EFD When cutting feed (feed per minute) is used in PMC axis control, the specification unit of feedrate data is:
0: Unchanged (1 times).
1: 100 times greater.

NOTE
When this parameter is set to 1, bit 3 (F10) of parameter No. 8002 is invalid.

#6 EZR In PMC axis control, bit 0 (ZRNx) of parameter No. 1005 is:
0: Invalid.
With a PMC controlled axis, the alarm PS0224, "ZERO RETURN NOT FINISHED" is not issued.
1: Valid.
A reference position return state check is made on a PMC controlled axis as with an NC axis according to the setting of bit 0 (ZRNx) of parameter No. 1005.

	#7	#6	#5	#4	#3	#2	#1	#0
8008								EMRx

[Input type] Parameter input
[Data type] Bit axis

#0 EMRx When a PMC axis control command is issued in mirror image state, the mirror image is:
0: Not considered.
1: Considered.
This parameter is valid in the mirror image mode set with the mirror image signals MI1 to MI8 <G106.0 to .7> set to "1" or bit 0 (MIRx) of parameter No. 12 set to 1.
If a movement is made along the same axis by doubly specifying a command with the CNC and PMC axis control when this parameter is set to 0, and the mirror image mode is set, a coordinate shift can occur afterwards. So, do not attempt to make such a movement.

8010	Selection of the DI/DO group for each axis controlled by the PMC

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 1 to 40
Specify the DI/DO group to be used to specify a command for each PMC-controlled axis.

For addresses of the fifth group and up, 1000 is added in steps of 4 groups.
For example:
The start address of the 10th group is G2154.
The start address of the 25th axis is G6142.

P8010	Description
1	DI/DO 1st group (G142 to G153) is used.
2	DI/DO 2nd group (G154 to G165) is used.
3	DI/DO 3rd group (G166 to G177) is used.
4	DI/DO 4th group (G178 to G189) is used.
5	DI/DO 5th group (G1142 to G1153) is used.
6	DI/DO 6th group (G1154 to G1165) is used.
:	:
13	DI/DO 13th group (G3142 to G3153) is used.
:	:
20	DI/DO 20th group (G4178 to G4189) is used.
21	DI/DO 21st group (G5142toG5153) is used.
:	:
29	DI/DO 29th group (G7142toG7153) is used.
:	:

P8010	Description
35	DI/DO 35th group (G8166toG8177) is used.
36	DI/DO 36th group (G8178toG8189) is used.
37	DI/DO 37th group (G9142toG9153) is used.
38	DI/DO 38th group (G9154toG9165) is used.
39	DI/DO 39th group (G9166toG9177) is used.
40	DI/DO 40th group (G9178toG9189) is used.

NOTE
When a value other than the above is set, the axis is not controlled by the PMC.

	#7	#6	#5	#4	#3	#2	#1	#0
8011								XRT

[Input type] Parameter input
[Data type] Bit axis

#0 XRT The axis that uses the group specified by parameter No. 8010 is:
0: Not controlled by the real time custom macro.
1: Controlled by the real time custom macro.

NOTE
1 This parameter is invalid for the axis for which 0 or a value outside the range is set by parameter No. 8010
2 When multiple axes are assigned to the same group by parameter No. 8010, these axes cannot be controlled by the real time custom macro. When multiple axes are assigned to the same group, be sure to set this bit to 0.
3 When this parameter No. 8011 is all 0s, the axis is used for PMC axis control.

8030	Time constant for exponential acceleration/deceleration in cutting feed or continuous feed under PMC axis control

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] msec
[Valid data range] 0 to 4000
For each axis, this parameter sets a time constant for exponential acceleration/deceleration in cutting feed or continuous feed under PMC axis control.

NOTE
When 0 is set in this parameter, the value set in parameter No. 1622 is used.
The value set in parameter No. 1622 is used also for linear acceleration/deceleration after cutting interpolation.

	#7	#6	#5	#4	#3	#2	#1	#0
8103							MWP	MWT

[Input type] Parameter input
[Data type] Bit

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 MWT As the signal interface for the waiting M code:
0: The path individual signal interface is used.
1: The path common signal interface is used.
This parameter can be selected only when 2-path control is used.

#1 MWP To specify a P command for the waiting M code/balance cut:
0: A binary value is used as conventionally done.
1: A path number combination is used.

	#7	#6	#5	#4	#3	#2	#1	#0
8107								ESB

[Input type] Parameter input
[Data type] Bit

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 ESB External subprogram calls on the data server:
0: Do not support multi-path operations.
1: Support multi-path operations.

NOTE
To use multi-path operations with external subprogram calls on the data server, set parameter No. 20 to 5.

8110	Waiting M code range (minimum value)
8111	Waiting M code range (maximum value)

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 ,100to99999999
A range of M code values can be set by specifying a minimum waiting M code value (parameter No. 8110) and a maximum waiting M code value (parameter No. 8111).
(parameter No. 8110) ≤ (waiting M code) ≤ (parameter No. 8111)
Set 0 in these parameters when the waiting M code is not used.

	#7	#6	#5	#4	#3	#2	#1	#0
8162						PKUx		

[Input type] Parameter input
[Data type] Bit axis

#2 PKUx In the parking state,
0: The absolute, relative, and machine coordinates are not updated.
1: The absolute and relative coordinates are updated. The machine coordinates are not updated.

NOTE
1 With an axis for which polar coordinate interpolation is specified, set this parameter to 1. If this parameter is set to 0, a coordinate shift can occur when a single block stop or feed hold is performed in the polar coordinate interpolation mode.
2 With an axis that is set to function as a synchronous master axis and synchronous slave axis at the same time (with bit 1 (SYWx) of parameter No. 8167), set this parameter to 1.
3 With an axis specified in the 3-dimensional coordinate conversion mode, set this parameter to 1. If this parameter is set to 0, the alarm PS0367, “3-D CONV. WAS COMMANDED IN SYNC MODE AS THE PARAMETER PKUx(NO.8162#2) IS 0.” is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
8163	NUMx							

[Input type] Parameter input
[Data type] Bit axis

#7 NUMx When neither synchronous control nor composite control is applied, a move command for the axis is:
0: Not disabled.
1: Disabled.

NOTE
If a move command is specified for an axis with NUMx set to 1 when neither synchronous control nor composite control is applied, alarm PS0353, “THE INSTRUCTION WAS DONE FOR THE AXIS WHICH WAS NOT ABLE TO MOVE.” is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
8167		SPVx	SWSx	SWMx	SGSx	SGMx	SYWx	
		SPVx					SYWx	

[Input type] Parameter input
[Data type] Bit axis

#1 SYWx The axis is:
0: Not used as a master axis and slave axis at the same time.
1: Used as a master axis and slave axis at the same time.

#2 SGMx In automatic workpiece coordinate system setting at the start of synchronous control, a tool offset is:
0: Considered.
1: Not considered.

NOTE
SGMx is enabled when bit 1 (SPMx) of parameter No. 8163 is set to 1.

#3 SGSx In automatic workpiece coordinate system setting at the end of synchronous control, a tool offset is:
0: Considered.
1: Not considered.

> **NOTE**
> SGSx is enabled when bit 2 (SPSx) of parameter No. 8163 or bit 6 (SPVx) of parameter No. 8167 is set to 1.

#4 SWMx In automatic workpiece coordinate system setting at the start of synchronous control, a workpiece shift is:
0: Not considered.
1: Considered.

> **NOTE**
> SWMx is enabled when bit 1 (SPMx) of parameter No. 8163 is set to 1.

#5 SWSx In automatic workpiece coordinate system setting at the end of synchronous control, a workpiece shift is:
0: Not considered.
1: Considered.

> **NOTE**
> SWSx is enabled when bit 2 (SPSx) of parameter No. 8163 or bit 6 (SPVx) of parameter No. 8167 is set to 1.

#6 SPVx At the end of synchronous control, automatic workpiece coordinate system setting for the slave axis is:
0: Not performed.
1: Performed.

> **NOTE**
> When a workpiece coordinate system is automatically set at the end of synchronous control, the workpiece coordinate system is calculated from the current machine coordinates and the workpiece coordinates for each axis at the reference position set in parameter No. 1250.

	#7	#6	#5	#4	#3	#2	#1	#0
8168			SFH					

[Input type] Parameter input
[Data type] Bit

#5 SFH For high-speed cycle cutting or high-speed binary program operation, superimposed control is:
0: Not applied.
1: Applied.

NOTE
When parameter SFH is set to 0 and superimposed control is applied for high-speed cycle cutting or high-speed binary program operation, alarm DS0070, “SUPERIMPOSE FOR HIGH-SPEED CYCLE CANNOT BE USED” is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
8169		SESx						

[Input type] Parameter input
[Data type] Bit axis

#6 SESx If a synchronization error is out of the tolerable range (specified with parameter No. 8181):
0: Alarm SV0407, “EXCESS ERROR”, is issued.
1: No alarm is issued. Instead, the excess synchronization error signal SEO<Fn559> is output.

SESx is valid when bit 1 (SERx) of parameter No. 8162 is 1. Specify the value of this parameter for the slave axis.

8180	Master axis with which an axis is synchronized under synchronous control

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 101, 102, 103, . . . , (path number)*100+(intra-path relative axis number) (101, 102, 103, . . . , 201, 202, 203, . . . , 1001, 1002, 1003, . . .)
This parameter sets the path number and intra-path relative axis number of the master axis with which each axis is synchronized. When zero is specified, the axis does not become a slave axis and is not synchronized with another axis. When an identical number is specified in two or more parameters, one master axis has two or more slave axes.

8183	Composite control axis of the other path in composite control for each axis

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 101, 102, 103, . . . , (path number)*100+(intra-path relative axis number) (101, 102, 103, . . . , 201, 202, 203, . . . , 1001, 1002, 1003, . . .)
This parameter sets with which axis each axis is to be placed under composite control. When zero is specified, control of the axis is not replaced under composite control. An identical number can be specified in two or more parameters, but composite control cannot be exercised for all of them at a time.

NOTE
When the two-path interface is used (bit 1 (MIX) of parameter No. 8166 is set to 1), set this parameter for path 2.

8186	Master axis under superimposed control

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 101, 102, 103, . . . , (path number)*100+(intra-path relative axis number) (101, 102, 103, . . . , 201, 202, 203, . . . , 1001, 1002, 1003, . . .)

This parameter sets the path number and intra-path relative axis number of a superimposed master axis for each axis when superimposed control is exercised. When zero is specified, the axis does not become a slave axis under superimposed control and the move pulse of another axis is not superimposed.
An identical number can be specified in two or more parameters to exercise superimposed control simultaneously. This means that superimposed control with one master axis and multiple slave axes is possible.
A slave axis may function as the master axis of another axis to allow three-generation superimposed control: parent (master axis) - child (slave axis/master axis) - grandchild (slave axis).

In this case, a movement along the child is made by its travel distance plus the travel distance of the parent, and a movement along the grandchild is made by its travel distance plus the travel distance of the child plus the travel distance of the parent.

Example of the relationship of parent (X1 of path 1) - child (X2 of path 2) - grandchild (X3 of path 3):
The travel distance of X1 is superimposed on X2, and the travel distances of X1 and X2 are further superimposed on X3.
Parameter No. 8186 (X axis) of path 2 = 101
Parameter No. 8186 (X axis) of path 3 = 201

	#7	#6	#5	#4	#3	#2	#1	#0
8200						AZR		

[Input type] Parameter input
[Data type] Bit path

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#2 AZR 0: The machine tool is moved along the Cartesian axis during manual reference position return along the slanted axis under angular axis control.
1: The machine tool is not moved along the Cartesian axis during manual reference position return along the slanted axis under angular axis control.

	#7	#6	#5	#4	#3	#2	#1	#0
8303				SYPx				

[Input type] Parameter input
[Data type] Bit axis

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#4 SYPx In axis synchronous control, some parameters must be set to the same value for the master and slave axes. When a value is set in such a parameter for the master axis:
0: The same value is not automatically set in the parameter for the slave axis.
1: The same value is automatically set in the parameter for the slave axis.

NOTE
1 For the parameters that can be set automatically, refer to Subsection 1.6.10, "Automatic Setting of Parameters for Slave Axes", in Connection Manual (Function) (B-64483EN-1).
2 Set this parameter to the same value for both the master and slave axes.

	#7	#6	#5	#4	#3	#2	#1	#0
8304			SCAx					

[Input type] Parameter input
[Data type] Bit axis

#5 SCAx In axis synchronous control:
0: Synchronous operation is performed when the axis synchronous control manual feed selection signal SYNCJ <G0140> or the axis synchronous control selection signal SYNC <G0138> for slave axes is set to 1.
1: Synchronous operation is performed at all times.
Set this parameter with a slave axis.

	#7	#6	#5	#4	#3	#2	#1	#0
8305						SRF		

[Input type] Parameter input
[Data type] Bit path

#2 SRF In axis synchronous control, G28, G30, and G53:
0: Make the same movement along the slave axis as a movement along the master axis.
1: Make movements along the slave axis and master axis independently to specified positions.

8314	Maximum allowable error in synchronization error check based on machine coordinates

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets a maximum allowable error in a synchronization error check based on machine coordinates. When the error between the master and slave axes in machine coordinates exceeds the value set in this parameter, the machine stops with the servo alarm SV0005, “SYNC EXCESS ERROR (MCN)”.
Set this parameter with a slave axis.

NOTE
Set 0 in this parameter when a synchronization error check is not made.

8323	Limit in positional deviation check in axis synchronous control

[Input type] Parameter input
[Data type] 2-word axis

[Unit of data] Detection unit
[Valid data range] 0 to 999999999
This parameter sets the maximum allowable difference between the master axis and slave axis position deviations. When the absolute value of a positional deviation difference exceeds the value set in this parameter in axis synchronous control, the alarm DS0001, "SYNC EXCESS ERROR (POS DEV)" is issued.
Set this parameter with a slave axis. If 0 is specified in this parameter, no position deviation difference check is made.

8325 **Maximum compensation value in synchronization establishment based on machine coordinates**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the maximum compensation value for synchronization. When a compensation value exceeding the value set in this parameter is detected, the servo alarm SV0001, "SYNC ALIGNMENT ERROR" is issued, and the synchronization establishment is not performed.
Specify a slave axis for this parameter. To enable this parameter, set the bit 7 (SOF) of parameter No. 8303 to 1. When 0 is set in this parameter, synchronization establishment is not performed.

8330 **Multiplier for a maximum allowable synchronization error immediately after power-up**

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 100
Until synchronization establishment is completed immediately after power-up, synchronization error excessive alarm 2 is checked using the maximum allowable error (parameter No. 8332) multiplied by the value set in this parameter.
If the result produced by multiplying the value of parameter No. 8332 by the value of this parameter exceeds 32767, the value is clamped to 32767.

8331 **Maximum allowable synchronization error for synchronization error excessive alarm 1**

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 32767
This parameter sets a maximum allowable synchronization error for synchronization error excessive alarm 1.
Set this parameter with a slave axis.

8332	Maximum allowable synchronization error for synchronization error excessive alarm 2

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 1 to 32767
This parameter sets a maximum allowable synchronization error for synchronization error excessive alarm 2.
Set this parameter with a slave axis.

8337	M code for turning off synchronization in axis synchronous control

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 999999999
This parameter specifies an M code for switching from synchronous operation to normal operation.
The M code set in this parameter is not buffered.

8338	M code for turning on synchronization in axis synchronous control

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 999999999
This parameter specifies an M code for switching from normal operation to synchronous operation.
The M code set in this parameter is not buffered.

8360								
								ROV

[Input type] Setting input
[Data type] Bit path

#0 ROV As rapid traverse override for a section from the chopping/oscillation start point to point R:
0: Chopping/oscillation override is used.
1: Rapid traverse override is used.

8370	Chopping/oscillation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes
This parameter sets which servo axis the chopping/oscillation axis corresponds to.

8371	Chopping/oscillation reference point (point R)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)

[Min. unit of data] Depend on the increment system of the chopping/oscillation axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
The data set in this parameter is absolute coordinates.

8372	Chopping/oscillation upper dead point

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the chopping/oscillation axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
The data set in this parameter is absolute coordinates.

8373	Chopping/oscillation lower dead point

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Min. unit of data] Depend on the increment system of the chopping/oscillation axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
The data set in this parameter is absolute coordinates.

8374	Chopping feedrate/oscillation base feedrate

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (input unit)
[Min. unit of data] Depend on the increment system of the chopping/oscillation axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the chopping feedrate/oscillation base feedrate.

8375	Maximum chopping/oscillation feedrate

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
The chopping/oscillation feedrate is clamped at this parameter setting. The maximum feedrate must be set for the chopping/oscillation axis. If this parameter is set to 0, no movement is made for chopping/oscillation.

NOTE
Please set a value that is smaller than the normal rapid traverse rate (parameter No. 1420) to Maximum chopping/oscillation feedrate.

8376	Chopping compensation factor

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 100
The value obtained by multiply the sum of the servo delay in an chopping operation and the acceleration/deceleration delay by the rate set in this parameter is used as chopping delay compensation. When this parameter is set to 0, chopping delay compensation is not applied.

8377	Chopping compensation start tolerance

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
In a chopping operation, compensation is applied when the difference between an amount of shortage at the upper dead point and that at the lower dead point due to the servo position control delay is less than the value set in this parameter. When this parameter is set to 0, compensation is not applied.

	#7	#6	#5	#4	#3	#2	#1	#0
8451	NOF			ZAG				

[Input type] Setting input
[Data type] Bit path

#4 ZAG The deceleration function based on cutting load in AI contour control (deceleration based on Z-axis fall angle) is:
0: Not performed.
1: Performed.
When this parameter is set to 1, be sure to set parameter Nos. 8456, 8457, and 8458.

#7 NOF In AI contour control, an F command is:
0: Not ignored.
1: Ignored.
When this parameter is set to 1, the specification of the maximum allowable feedrate set in parameter No. 8465 is assumed.

8456	Override for range 2 that is applied during deceleration according to the cutting load in AI contour control
8457	Override for range 3 that is applied during deceleration according to the cutting load in AI contour control
8458	Override for range 4 that is applied during deceleration according to the cutting load in AI contour control

[Input type] Setting input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 100

For the function of decelerating according to the cutting load in AI contour control, the override set in a parameter can be applied according to the angle at which the tool moves downward along the Z-axis. The feedrate obtained according to other conditions is multiplied by the override for the range containing angle θ at which the tool moves downward.
However, when bit 1 (ZG2) of parameter No. 19515 is set to 0, no parameter is available to range 1, and 100% is applied at all times. When bit 1 (ZG2) of parameter No. 19515 is set to 1, set an override value for range 1 in parameter No. 19516.

Range 1 $0° \leq \theta < 30°$
Range 2 $30° \leq \theta < 45°$
Range 3 $45° \leq \theta < 60°$
Range 4 $60° \leq \theta \leq 90°$

8465 **Maximum allowable feedrate for AI contour control**

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the maximum allowable feedrate for contour control.
If a feedrate higher than the setting of this parameter is specified in the AI contour control mode, the feedrate is clamped to that set in this parameter.
If this parameter is set to 0, no clamping is performed.
When bit 7 (NOF) of parameter No. 8451 is set to 1, the tool moves, assuming that the feedrate set in this parameter is specified. If 0 is set in this parameter at this time, a movement is made at the specified feedrate.

8466 **Maximum allowable feedrate for AI contour control (when a rotation axis is singly specified)**

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
This parameter sets the maximum allowable feedrate for AI contour control when a rotation axis is singly specified.
If a feedrate higher than the setting of this parameter is specified in the AI contour control mode, the feedrate is clamped to that set in this parameter.
If this parameter is set to 0, the feedrate is clamped to that set in parameter No. 8465.
When bit 7 (NOF) of parameter No. 8451 is set to 1 and a rotation axis is singly specified, the tool moves, assuming that the feedrate set in this parameter is specified. If 0 is set in this parameter at this time, the tool moves at the feedrate specified in parameter No. 8465.

8486 **Maximum travel distance of a block where smooth interpolation or Nano smoothing is applied**

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter specifies a block length used as a reference to decide whether to apply smooth interpolation or Nano smoothing. If the line specified in a block is longer than the value set in the parameter, smooth interpolation or Nano smoothing is not applied to that block.

8487	Angle at which smooth interpolation or Nano smoothing is turned off

[Input type] Setting input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 to 90
This parameter sets the angle used to determine whether to apply smooth interpolation or Nano smoothing.
At a point having a difference in angle greater than this setting, smooth interpolation or Nano smoothing is turned off.

8490	Minimum travel distance of a block where smooth interpolation or Nano smoothing is applied

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets a block length used to determine whether to apply smooth interpolation or Nano smoothing.
If the line specified in a block is shorter than the value set in this parameter, smooth interpolation or Nano smoothing is not applied to that block.

	#7	#6	#5	#4	#3	#2	#1	#0
8900								PWE

[Input type] Setting input
[Data type] Bit

#0 PWE The setting, from an external device and MDI panel, of those parameters that cannot be set by setting input is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
9033			SHS					

[Input type] Parameter input
[Data type] Bit path

#5 SHS When the high-speed cycle cutting function is enabled, variables #20000 and after are treated as:
0: High-speed cycle cutting variables.
1: P-CODE variables.

	#7	#6	#5	#4	#3	#2	#1	#0
10335								MSC

[Input type] Parameter input
[Data type] Bit path

#0 MSC The reconfirming of midway block start of operator error prevent function is:
0: Enabled independently for each path.
1: Enabled for the local path and those paths for this parameter is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
10345							L2D	

[Input type] Parameter input
[Data type] Bit

#1 L2D When the forbidden area of the stored stroke check 2, 3 (Parameter No.1322, 1323) is set or it is changed by G22 command, the setting value for axes with diameter specification is:
0: Half of command value
1: Command value

	#7	#6	#5	#4	#3	#2	#1	#0
10350	AOS							

[Input type] Parameter input
[Data type] Bit

#7 AOS When the number of control paths in a system is 2 or more, the high-speed cycle cutting function is:
0: Not executed.
1: Executed.

	#7	#6	#5	#4	#3	#2	#1	#0
10410								NRTx

[Input type] Parameter input
[Data type] Bit axis

#0 NRTx In tool retract and recover or manual intervention and return, the axis is:
0: Subject to tool retract and recover or manual intervention and return.
1: Not subject to tool retract and recover or manual intervention and return.

10461	RGB value of color palette 1 for text for color set 3
10462	RGB value of color palette 2 for text for color set 3
to	to
10475	RGB value of color palette 15 for text for color set 3

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0 to 151515
Each of these parameters sets the RGB value of each color palette for text by specifying a 6-digit number as described below.
rrggbb: 6-digit number (rr: red data, gg: green data, bb: blue data)

The valid data range of each color is 0 to 15 (same as the tone levels on the color setting screen). When a number equal to or greater than 16 is specified, the specification of 15 is assumed.

[Example] When the tone level of a color is: red:1 green:2, blue:3, set 10203 in the parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
10485								STC

[Input type] Setting input
[Data type] Bit path

#0 STC If, in a TCP start block (G43.4, G43.5), address "L" is omitted, TCP is:
0: Started as normal TCP.
1: Started as High-speed Smooth TCP (Rotation axes compensation).

10486	First rotation axis compensation tolerance in High-speed Smooth TCP mode (G43.4L1, G43.5L1)

[Input type] Setting input
[Data type] Real path
[Unit of data] degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
If the first rotation axis is compensated in Rotation axes compensation (G43.4L1, G43.5L1), the maximum change from the specified value before compensation is limited by this setting.
The first rotation axis is the axis specified for parameter No. 19681.
If 0 is set in this parameter, the first rotation axis is not compensated.
If a value other than 0 is set in parameter No. 10490, this parameter is clamped with the value of parameter No. 10490 as the upper limit.

10487	Second rotation axis compensation tolerance in High-speed Smooth TCP mode (G43.4L1, G43.5L1)

[Input type] Setting input
[Data type] Real path
[Unit of data] degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
If the second rotation axis is compensated in Rotation axes compensation (G43.4L1, G43.5L1), the maximum change from the specified value before compensation is limited by this setting.
The second rotation axis is the axis specified for parameter No. 19686.
If 0 is set in this parameter, the second rotation axis is not compensated.
If a value other than 0 is set in parameter No. 10491, this parameter is clamped with the value of parameter No. 10491 as the upper limit.

10490	Maximum first rotation axis compensation tolerance in High-speed Smooth TCP mode (G43.4L1, G43.5L1)

[Input type] Parameter input
[Data type] Real path
[Unit of data] degree

[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
If a value other than 0 is set in this parameter and the setting of parameter No. 10486 is equal to or greater than the setting of this parameter, the setting of this parameter is regarded as the setting of parameter No. 10486.
If 0 is set in this parameter, parameter No. 10486 is not limited.

10491	Maximum second rotation axis compensation tolerance in High-speed Smooth TCP mode (G43.4L1, G43.5L1)

[Input type] Parameter input
[Data type] Real path
[Unit of data] degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
If a value other than 0 is set in this parameter and the setting of parameter No. 10487 is equal to or greater than the setting of this parameter, the setting of this parameter is regarded as the setting of parameter No. 10487.
If 0 is set in this parameter, parameter No. 10487 is not limited.

	#7	#6	#5	#4	#3	#2	#1	#0
10930	TFU	ENO	IIA	ICN	ICV	ICT	ICD	ICE

[Input type] Parameter input
[Data type] Bit

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 ICE Built-in 3D interference check function is:
0: Enabled.
1: Disabled.
However, the setting related to 3D interference check is effective.
This parameter can disable built-in 3D interference check function. This parameter allows 3D interference check setting to be made while a movement is being made on an axis, even before the setting of built-in 3D interference check function is not completed.

NOTE
With a machine that uses built-in 3D interference check function, set ICE = 0 usually. If ICE = 1, the alarm (PS0494) “3DCHK FUNCTION INVALID” is issued at power-on time. This alarm can be reset by pressing the MDI keys “CAN” and “RESET”.

#1 ICD Built-in 3D interference check setting screen is:
0: Displayed.
1: Not displayed.

#2 ICT The method for built-in 3D interference check function to find tool offset number changed is:
0: The PMC window (function code 431)
1: The tool management function with the PMC window (function code 329)

#3 ICV In built-in 3D interference function, when hen 0 is specified for the figure number of valid figure of 3D interference check:
0: Figure 1 is effective.
1: No figure. (Removed from interference check target)

#4 ICN In the method of notifying the tool change for built-in 3D interference function, which is specified by parameter ICT (No.10930#2), when 0 is specified for the tool offset number, or the first spindle number of tool management function:
0: Tool figure and tool-holder figure are not changed.
1: The tool is removed from the interference check target, and tool holder figure follows parameter No.10960-10963.

#5 IIA In built-in 3D interference function, in case parameter ICT(No.10930#2) is 0, if the specified tool offset number is invalid:
0: Alarm PS0492, "3DCHK FIG. ILLEGAL: [Target name]", is issued.
(The figure of interference check target is invalid.)
1: Alarm PS0492 is not issued and tool figure and tool-holder figure are not changed.
It can be confirmed whether figure is invalid by the completion code of the PMC window (function code 431).

#6 ENO In built-in 3D interference check function, when the number of controlled paths is 1, the number of object is:
0: Three.
1: Six.

#7 TFU The update function of the tool figure by built-in 3D interference check setting change signal TDICHG <G519.4> is:
0: Invalid.
1: Valid.

	#7	#6	#5	#4	#3	#2	#1	#0
10931	TDIC107	TDIC106	TDIC105	TDIC104	TDIC103	TDIC102	TDIC101	TDIC100

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIC100 In built-in 3D interference check function, check for interference between tool 1 and tool holder 1 is:
0: Disabled.
1: Enabled.

#1 TDIC101 In built-in 3D interference check function, check for interference between tool 1 and object 1 is:
0: Enabled.
1: Disabled.

#2 TDIC102 In built-in 3D interference check function, check for interference between tool 1 and object 2 is:
0: Enabled.
1: Disabled.

#3 TDIC103 In built-in 3D interference check function, check for interference between tool 1 and object 3 is:
0: Enabled.
1: Disabled.

#4 TDIC104 In built-in 3D interference check function, check for interference between tool holder 1 and object 1 is:
0: Enabled.
1: Disabled.

#5 TDIC105 In built-in 3D interference check function, check for interference between tool holder 1 and object 2 is:
0: Enabled.
1: Disabled.

#6 TDIC106 In built-in 3D interference check function, check for interference between tool holder 1 and object 3 is:
0: Enabled.
1: Disabled.

#7 TDIC107 In built-in 3D interference check function, check for interference between object 1 and object 2 is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10932							TDIC109	TDIC108

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIC108 In built-in 3D interference check function, check for interference between object 1 and object 3 is:
0: Disabled.
1: Enabled.

#1 TDIC109 In built-in 3D interference check function, check for interference between object 2 and object 3 is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10933	TDIC207	TDIC206	TDIC205	TDIC204	TDIC203	TDIC202	TDIC201	TDIC200

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to "1".

#0 TDIC200 In built-in 3D interference check function, check for interference between tool 1 and tool object 4 is:
0: Enabled.
1: Disabled.

#1 TDIC201 In built-in 3D interference check function, check for interference between tool 1 and object 5 is:
0: Enabled.
1: Disabled.

#2 TDIC202 In built-in 3D interference check function, check for interference between tool 1 and object 6 is:
0: Enabled.
1: Disabled.

#3 TDIC203 In built-in 3D interference check function, check for interference between tool holder 1 and object 4 is:
0: Enabled.
1: Disabled.

#4 TDIC204 In built-in 3D interference check function, check for interference between tool holder 1 and object 5 is:
0: Enabled.
1: Disabled.

#5 TDIC205 In built-in 3D interference check function, check for interference between tool holder 1 and object 6 is:
0: Enabled.
1: Disabled.

#6 TDIC206 In built-in 3D interference check function, check for interference between object 1 and object 4 is:
0: Disabled.
1: Enabled.

#7 TDIC207 In built-in 3D interference check function, check for interference between object 1 and object 5 is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10934	TDIC215	TDIC214	TDIC213	TDIC212	TDIC211	TDIC210	TDIC209	TDIC208

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIC208 In built-in 3D interference check function, check for interference between object 1 and object 6 is:
0: Disabled.
1: Enabled.

#1 TDIC209 In built-in 3D interference check function, check for interference between object 2 and object 4 is:
0: Disabled.
1: Enabled.

#2 TDIC210 In built-in 3D interference check function, check for interference between object 2 and object 5 is:
0: Disabled.
1: Enabled.

#3 TDIC211 In built-in 3D interference check function, check for interference between object 2 and object 6 is:
0: Disabled.
1: Enabled.

#4 TDIC212 In built-in 3D interference check function, check for interference between object 3 and object 4 is:
0: Disabled.
1: Enabled.

#5 TDIC213 In built-in 3D interference check function, check for interference between object 3 and object 5 is:
0: Disabled.
1: Enabled.

#6 TDIC214 In built-in 3D interference check function, check for interference between object 3 and object 6 is:
0: Disabled.
1: Enabled.

#7 TDIC215 In built-in 3D interference check function, check for interference between object 4 and object 5 is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10935	TDIC223	TDIC222	TDIC221	TDIC220	TDIC219	TDIC218	TDIC217	TDIC216

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIC216 In built-in 3D interference check function, check for interference between object 4 and object 6 is:
0: Disabled.
1: Enabled.

#1 TDIC217 In built-in 3D interference check function, check for interference between object 5 and object 6 is:
0: Disabled.
1: Enabled.

#2 TDIC218 In built-in 3D interference check function, check for interference between tool 1 and tool 2 is:
0: Enabled.
1: Disabled.

#3 TDIC219 In built-in 3D interference check function, check for interference between tool 1 and tool holder 2 is:
0: Enabled.
1: Disabled.

#4 TDIC220 In built-in 3D interference check function, check for interference between tool holder 1 and tool 2 is:
0: Enabled.
1: Disabled.

#5 TDIC221 In built-in 3D interference check function, check for interference between tool holder 1 and tool holder 2 is:
0: Enabled.
1: Disabled.

#6 TDIC222 In built-in 3D interference check function, check for interference between object 1 and tool 2 is:
0: Enabled.
1: Disabled.

#7 TDIC223 In built-in 3D interference check function, check for interference between object 1 and tool holder 2 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10936	TDIC231	TDIC230	TDIC229	TDIC228	TDIC227	TDIC226	TDIC225	TDIC224

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIC224 In built-in 3D interference check function, check for interference between object 2 and tool 2 is:
0: Enabled.
1: Disabled.

#1 TDIC225 In built-in 3D interference check function, check for interference between object 2 and tool holder 2 is:
0: Enabled.
1: Disabled.

#2 TDIC226 In built-in 3D interference check function, check for interference between object 3 and tool 2 is:
0: Enabled.
1: Disabled.

#3 TDIC227 In built-in 3D interference check function, check for interference between object 3 and tool holder 2 is:
0: Enabled.
1: Disabled.

#4 TDIC228 In built-in 3D interference check function, check for interference between object 4 and tool 2 is:
0: Enabled.
1: Disabled.

#5 TDIC229 In built-in 3D interference check function, check for interference between object 4 and tool holder 2 is:
0: Enabled.
1: Disabled.

#6 TDIC230 In built-in 3D interference check function, check for interference between object 5 and tool 2 is:
0: Enabled.
1: Disabled.

#7 TDIC231 In built-in 3D interference check function, check for interference between object 5 and tool holder 2 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10937						TDIC234	TDIC233	TDIC232

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to "1".

#0 TDIC232 In built-in 3D interference check function, check for interference between object 6 and tool 2 is:
0: Enabled.
1: Disabled.

#1 TDIC233 In built-in 3D interference check function, check for interference between object 6 and tool holder 2 is:
0: Enabled.
1: Disabled.

#2 TDIC234 In built-in 3D interference check function, check for interference between tool 2 and tool holder 2 is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10938	TDIC307	TDIC306	TDIC305	TDIC304	TDIC303	TDIC302	TDIC301	TDIC300

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to "1".

#0 TDIC300 In built-in 3D interference check function, check for interference between tool 1 and tool 3 is:
0: Enabled.
1: Disabled.

#1 TDIC301 In built-in 3D interference check function, check for interference between tool 1 and tool holder 3 is:
0: Enabled.
1: Disabled.

#2 TDIC302 In built-in 3D interference check function, check for interference between tool holder 1 and tool 3 is:
0: Enabled.
1: Disabled.

#3 TDIC303 In built-in 3D interference check function, check for interference between tool holder 1 and tool holder 3 is:
0: Enabled.
1: Disabled.

#4 TDIC304 In built-in 3D interference check function, check for interference between object 1 and tool 3 is:
0: Enabled.
1: Disabled.

#5 TDIC305 In built-in 3D interference check function, check for interference between object 1 and tool holder 3 is:
0: Enabled.
1: Disabled.

#6 TDIC306 In built-in 3D interference check function, check for interference between object 2 and tool 3 is:
0: Enabled.
1: Disabled.

#7 TDIC307 In built-in 3D interference check function, check for interference between object 2 and tool holder 3 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10939	TDIC315	TDIC314	TDIC313	TDIC312	TDIC311	TDIC310	TDIC309	TDIC308

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIC308 In built-in 3D interference check function, check for interference between object 3 and tool 3 is:
0: Enabled.
1: Disabled.

#1 TDIC309 In built-in 3D interference check function, check for interference between object 3 and tool holder 3 is:
0: Enabled.
1: Disabled.

#2 TDIC310 In built-in 3D interference check function, check for interference between object 4 and tool 3 is:
0: Enabled.
1: Disabled.

#3 TDIC311 In built-in 3D interference check function, check for interference between object 4 and tool holder 3 is:
0: Enabled.
1: Disabled.

#4 TDIC312 In built-in 3D interference check function, check for interference between object 5 and tool 3 is:
0: Enabled.
1: Disabled.

#5 TDIC313 In built-in 3D interference check function, check for interference between object 5 and tool holder 3 is:
0: Enabled.
1: Disabled.

#6 TDIC314 In built-in 3D interference check function, check for interference between object 6 and tool 3 is:
0: Enabled.
1: Disabled.

#7 TDIC315 In built-in 3D interference check function, check for interference between object 6 and tool holder 3 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10940				TDIC320	TDIC319	TDIC318	TDIC317	TDIC316

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIC316 In built-in 3D interference check function, check for interference between tool 2 and tool 3 is:
0: Enabled.
1: Disabled.

#1 TDIC317 In built-in 3D interference check function, check for interference between tool 2 and tool holder 3 is:
0: Enabled.
1: Disabled.

#2 TDIC318 In built-in 3D interference check function, check for interference between tool holder 2 and tool 3 is:
0: Enabled.
1: Disabled.

#3 TDIC319 In built-in 3D interference check function, check for interference between tool holder 2 and tool holder 3 is:
0: Enabled.
1: Disabled.

#4 TDIC320 In built-in 3D interference check function, check for interference between tool 3 and tool holder 3 is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10941	TDIC407	TDIC406	TDIC405	TDIC404	TDIC403	TDIC402	TDIC401	TDIC400

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIC400 In built-in 3D interference check function, check for interference between tool 1 and tool 4 is:
0: Enabled.
1: Disabled.

#1 TDIC401 In built-in 3D interference check function, check for interference between tool 1 and tool holder 4 is:
0: Enabled.
1: Disabled.

#2 TDIC402 In built-in 3D interference check function, check for interference between tool holder 1 and tool 4 is:
0: Enabled.
1: Disabled.

#3 TDIC403 In built-in 3D interference check function, check for interference between tool holder 1 and tool holder 4 is:
0: Enabled.
1: Disabled.

#4 TDIC404 In built-in 3D interference check function, check for interference between object 1 and tool 4 is:
0: Enabled.
1: Disabled.

#5 TDIC405 In built-in 3D interference check function, check for interference between object 1 and tool holder 4 is:
0: Enabled.
1: Disabled.

#6 TDIC406 In built-in 3D interference check function, check for interference between object 2 and tool 4 is:
0: Enabled.
1: Disabled.

#7 TDIC407 In built-in 3D interference check function, check for interference between object 2 and tool holder 4 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10942	TDIC415	TDIC414	TDIC413	TDIC412	TDIC411	TDIC410	TDIC409	TDIC408

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIC408 In built-in 3D interference check function, check for interference between object 3 and tool 4 is:
0: Enabled.
1: Disabled.

#1 TDIC409 In built-in 3D interference check function, check for interference between object 3 and tool holder 4 is:
0: Enabled.
1: Disabled.

#2 TDIC410 In built-in 3D interference check function, check for interference between object 4 and tool 4 is:
0: Enabled.
1: Disabled.

#3 TDIC411 In built-in 3D interference check function, check for interference between object 4 and tool holder 4 is:
0: Enabled.
1: Disabled.

#4 TDIC412 In built-in 3D interference check function, check for interference between object 5 and tool 4 is:
0: Enabled.
1: Disabled.

#5 TDIC413 In built-in 3D interference check function, check for interference between object 5 and tool holder 4 is:
0: Enabled.
1: Disabled.

#6 TDIC414 In built-in 3D interference check function, check for interference between object 6 and tool 4 is:
0: Enabled.
1: Disabled.

#7 TDIC415 In built-in 3D interference check function, check for interference between object 6 and tool holder 4 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10943	TDIC423	TDIC422	TDIC421	TDIC420	TDIC419	TDIC418	TDIC417	TDIC416

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIC416 In built-in 3D interference check function, check for interference between tool 2 and tool 4 is:
0: Enabled.
1: Disabled.

#1 TDIC417 In built-in 3D interference check function, check for interference between tool 2 and tool holder 4 is:
0: Enabled.
1: Disabled.

#2 TDIC418 In built-in 3D interference check function, check for interference between tool holder 2 and tool 4 is:
0: Enabled.
1: Disabled.

#3 TDIC419 In built-in 3D interference check function, check for interference between tool holder 2 and tool holder 4 is:
0: Enabled.
1: Disabled.

#4 TDIC420 In built-in 3D interference check function, check for interference between tool 3 and tool 4 is:
0: Enabled.
1: Disabled.

#5 TDIC421 In built-in 3D interference check function, check for interference between tool 3 and tool holder 4 is:
0: Enabled.
1: Disabled.

#6 TDIC422 In built-in 3D interference check function, check for interference between tool holder 3 and tool 4 is:
0: Enabled.
1: Disabled.

#7 TDIC423 In built-in 3D interference check function, check for interference between tool holder 3 and tool holder 4 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10944								TDIC424

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIC424 In built-in 3D interference check function, check for interference between tool 4 1 and tool holder 4 is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10945	TDIR107	TDIR106	TDIR105	TDIR104	TDIR103	TDIR102	TDIR101	TDIR100

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIR100 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 1 and tool holder 1 is:
0: Enabled.
1: Disabled.

#1 TDIR101 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 1 and object 1 is:
0: Enabled.
1: Disabled.

#2 TDIR102 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 1 and object 2 is:
0: Enabled.
1: Disabled.

#3 TDIR103 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 1 and object 3 is:
0: Enabled.
1: Disabled.

#4 TDIR104 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool holder 1 and object 1 is:
0: Enabled.
1: Disabled.

#5 TDIR105 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool holder 1 and object 2 is:
0: Enabled.
1: Disabled.

#6 TDIR106 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool holder 1 and object 3 is:
0: Enabled.
1: Disabled.

#7 TDIR107 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 1 and object 2 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10946							TDIR109	TDIR108

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIR108 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 1 and object 3 is:
0: Enabled.
1: Disabled.

#1 TDIR109 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 2 and object 3 is:

0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10947	TDIR207	TDIR206	TDIR205	TDIR204	TDIR203	TDIR202	TDIR201	TDIR200

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIR200 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 1 and tool object 4 is:

0: Enabled.
1: Disabled.

#1 TDIR201 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 1 and object 5 is:

0: Enabled.
1: Disabled.

#2 TDIR202 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 1 and object 6 is:

0: Enabled.
1: Disabled.

#3 TDIR203 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool holder 1 and object 4 is:

0: Enabled.
1: Disabled.

#4 TDIR204 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool holder 1 and object 5 is:

0: Enabled.
1: Disabled.

#5 TDIR205 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool holder 1 and object 6 is:

0: Enabled.
1: Disabled.

#6 TDIR206 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 1 and object 4 is:
0: Enabled.
1: Disabled.

#7 TDIR207 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 1 and object 5 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10948	TDIR215	TDIR214	TDIR213	TDIR212	TDIR211	TDIR210	TDIR209	TDIR208

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIR208 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 1 and object 6 is:
0: Enabled.
1: Disabled.

#1 TDIR209 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 2 and object 4 is:
0: Enabled.
1: Disabled.

#2 TDIR210 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 2 and object 5 is:
0: Enabled.
1: Disabled.

#3 TDIR211 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 2 and object 6 is:
0: Enabled.
1: Disabled.

#4 TDIR212 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 3 and object 4 is:
0: Enabled.
1: Disabled.

#5 TDIR213 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 3 and object 5 is:
0: Enabled.
1: Disabled.

#6 TDIR214 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 3 and object 6 is:
0: Enabled.
1: Disabled.

#7 TDIR215 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 4 and object 5 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10949	TDIR223	TDIR222	TDIR221	TDIR220	TDIR219	TDIR218	TDIR217	TDIR216

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIR216 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 4 and object 6 is:
0: Enabled.
1: Disabled.

#1 TDIR217 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 5 and object 6 is:
0: Enabled.
1: Disabled.

#2 TDIR218 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 1 and tool 2 is:
0: Enabled.
1: Disabled.

#3 TDIR219 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 1 and tool holder 2 is:
0: Enabled.
1: Disabled.

#4 TDIR220 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = "1", check for interference between tool holder 1 and tool 2 is:
0: Enabled.
1: Disabled.

#5 TDIR221 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = "1", check for interference between tool holder 1 and tool holder 2 is:
0: Enabled.
1: Disabled.

#6 TDIR222 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = "1", check for interference between object 1 and tool 2 is:
0: Enabled.
1: Disabled.

#7 TDIR223 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = "1", check for interference between object 1 and tool holder 2 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10950	TDIR231	TDIR230	TDIR229	TDIR228	TDIR227	TDIR226	TDIR225	TDIR224

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to "1".

#0 TDIR224 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = "1", check for interference between object 2 and tool 2 is:
0: Enabled.
1: Disabled.

#1 TDIR225 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = "1", check for interference between object 2 and tool holder 2 is:
0: Enabled.
1: Disabled.

#2 TDIR226 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = "1", check for interference between object 3 and tool 2 is:
0: Enabled.
1: Disabled.

#3 TDIR227 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 3 and tool holder 2 is:
0: Enabled.
1: Disabled.

#4 TDIR228 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 4 and tool 2 is:
0: Enabled.
1: Disabled.

#5 TDIR229 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 4 and tool holder 2 is:
0: Enabled.
1: Disabled.

#6 TDIR230 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 5 and tool 2 is:
0: Enabled.
1: Disabled.

#7 TDIR231 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 5 and tool holder 2 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10951						TDIR234	TDIR233	TDIR232

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIR232 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 6 and tool 2 is:
0: Enabled.
1: Disabled.

#1 TDIR233 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 6 and tool holder 2 is:
0: Enabled.
1: Disabled.

#2 TDIR234 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 2 and tool holder 2 is:

0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10952	TDIR307	TDIR306	TDIR305	TDIR304	TDIR303	TDIR302	TDIR301	TDIR300

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIR300 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 1 and tool 3 is:

0: Enabled.
1: Disabled.

#1 TDIR301 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 1 and tool holder 3 is:

0: Enabled.
1: Disabled.

#2 TDIR302 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool holder 1 and tool 3 is:

0: Enabled.
1: Disabled.

#3 TDIR303 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool holder 1 and tool holder 3 is:

0: Enabled.
1: Disabled.

#4 TDIR304 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 1 and tool 3 is:

0: Enabled.
1: Disabled.

#5 TDIR305 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 1 and tool holder 3 is:

0: Enabled.
1: Disabled.

#6 TDIR306 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 2 and tool 3 is:
0: Enabled.
1: Disabled.

#7 TDIR307 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 2 and tool holder 3 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10953	TDIR315	TDIR314	TDIR313	TDIR312	TDIR311	TDIR310	TDIR309	TDIR308

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIR308 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 3 and tool 3 is:
0: Enabled.
1: Disabled.

#1 TDIR309 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 3 and tool holder 3 is:
0: Enabled.
1: Disabled.

#2 TDIR310 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 4 and tool 3 is:
0: Enabled.
1: Disabled.

#3 TDIR311 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 4 and tool holder 3 is:
0: Enabled.
1: Disabled.

#4 TDIR312 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 5 and tool 3 is:
0: Enabled.
1: Disabled.

#5 TDIR313 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 5 and tool holder 3 is:
0: Enabled.
1: Disabled.

#6 TDIR314 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 6 and tool 3 is:
0: Enabled.
1: Disabled.

#7 TDIR315 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 6 and tool holder 3 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10954				TDIR320	TDIR319	TDIR318	TDIR317	TDIR316

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIR316 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 2 and tool 3 is:
0: Enabled.
1: Disabled.

#1 TDIR317 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 2 and tool holder 3 is:
0: Enabled.
1: Disabled.

#2 TDIR318 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool holder 2 and tool 3 is:
0: Enabled.
1: Disabled.

#3 TDIR319 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool holder 2 and tool holder 3 is:
0: Enabled.
1: Disabled.

#4 TDIR320 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 3 and tool holder 3 is:

0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10955	TDIR407	TDIR406	TDIR405	TDIR404	TDIR403	TDIR402	TDIR401	TDIR400

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIR400 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 1 and tool 4 is:

0: Enabled.
1: Disabled.

#1 TDIR401 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 1 and tool holder 4 is:

0: Enabled.
1: Disabled.

#2 TDIR402 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool holder 1 and tool 4 is:

0: Enabled.
1: Disabled.

#3 TDIR403 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool holder 1 and tool holder 4 is:

0: Enabled.
1: Disabled.

#4 TDIR404 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 1 and tool 4 is:

0: Enabled.
1: Disabled.

#5 TDIR405 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 1 and tool holder 4 is:

0: Enabled.
1: Disabled.

#6 TDIR406 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = "1", check for interference between object 2 and tool 4 is:
0: Enabled.
1: Disabled.

#7 TDIR407 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = "1", check for interference between object 2 and tool holder 4 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10956	TDIR415	TDIR414	TDIR413	TDIR412	TDIR411	TDIR410	TDIR409	TDIR408

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to "1".

#0 TDIR408 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = "1", check for interference between object 3 and tool 4 is:
0: Enabled.
1: Disabled.

#1 TDIR409 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = "1", check for interference between object 3 and tool holder 4 is:
0: Enabled.
1: Disabled.

#2 TDIR410 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = "1", check for interference between object 4 and tool 4 is:
0: Enabled.
1: Disabled.

#3 TDIR411 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = "1", check for interference between object 4 and tool holder 4 is:
0: Enabled.
1: Disabled.

#4 TDIR412 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = "1", check for interference between object 5 and tool 4 is:
0: Enabled.
1: Disabled.

#5 TDIR413 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 5 and tool holder 4 is:
0: Enabled.
1: Disabled.

#6 TDIR414 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 6 and tool 4 is:
0: Enabled.
1: Disabled.

#7 TDIR415 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between object 6 and tool holder 4 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10957	TDIR423	TDIR422	TDIR421	TDIR420	TDIR419	TDIR418	TDIR417	TDIR416

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIR416 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 2 and tool 4 is:
0: Enabled.
1: Disabled.

#1 TDIR417 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 2 and tool holder 4 is:
0: Enabled.
1: Disabled.

#2 TDIR418 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool holder 2 and tool 4 is:
0: Enabled.
1: Disabled.

#3 TDIR419 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool holder 2 and tool holder 4 is:
0: Enabled.
1: Disabled.

#4 TDIR420 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 3 and tool 4 is:
0: Enabled.
1: Disabled.

#5 TDIR421 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool 3 and tool holder 4 is:
0: Enabled.
1: Disabled.

#6 TDIR422 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool holder 3 and tool 4 is:
0: Enabled.
1: Disabled.

#7 TDIR423 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool holder 3 and tool holder 4 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10958								TDIR424

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIR424 In built-in 3D interference check function, during cutting feed, canned cycle or 3D interference check between specified targets disable signal (TDISD) = “1”, check for interference between tool4 1 and tool holder 4 is:
0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
10959								TTD

[Input type] Parameter input
[Data type] Bit

#0 TTD In built-in 3D interference check function, when the tool of the tip figure is automatically made, “tool setting” data in the tool geometry size data is:
0: Not considered.
1: Considered.

10960	Figure number of tool holder 1 in built-in 3D interference check function

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to the number of tool holder 1 figures
In built-in 3D interference check function, the figure number of tool holder 1 is specified when tool 1 is removed from the interference check target by means that 0 is specified for the tool offset number, or the first spindle number of tool management function.
When this parameter is 0, tool holder 1 is removed from the interference check target.
This parameter is active when parameter ICN (No.10930#4) is 1.

10961	Figure number of tool holder 2 in built-in 3D interference check function

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to the number of tool holder 2 figures
In built-in 3D interference check function, the figure number of tool holder 2 is specified when tool 2 is removed from the interference check target by means that 0 is specified for the tool offset number, or the first spindle number of tool management function.
When this parameter is 0, tool holder 2 is removed from the interference check target.
This parameter is active when parameter ICN (No.10930#4) is 1.

10962	Figure number of tool holder 3 in built-in 3D interference check function

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to the number of tool holder 3 figures
In built-in 3D interference check function, the figure number of tool holder 3 is specified when tool 3 is removed from the interference check target by means that 0 is specified for the tool offset number, or the first spindle number of tool management function.
When this parameter is 0, tool holder 3 is removed from the interference check target.
This parameter is active when parameter ICN (No.10930#4) is 1.

10963	Figure number of tool holder 4 in built-in 3D interference check function

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to the number of tool holder 4 figures
In built-in 3D interference check function, the figure number of tool holder 4 is specified when tool 4 is removed from the interference check target by means that 0 is specified for the tool offset number, or the first spindle number of tool management function.
When this parameter is 0, tool holder 4 is removed from the interference check target.
This parameter is active when parameter ICN (No.10930#4) is 1.

10965	Margin width in built-in 3D interference check

[Input type] Parameter input
[Data type] Real
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
In built-in 3D interference check function, set interference check margin width. If this parameter is set to 0, margin width is automatically calculated according to the interference check processing time. If this parameter is set to negative value, margin width becomes 0.

	#7	#6	#5	#4	#3	#2	#1	#0
10966	TDISH14	TDIST14	TDISH13	TDIST13	TDISH12	TDIST12	TDISH11	TDIST11

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIST11 In built-in 3D interference check function, when tool 1 and object 1 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#1 TDISH11 In built-in 3D interference check function, when tool holder 1 and object 1 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#2 TDIST12 In built-in 3D interference check function, when tool 2 and object 1 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#3 TDISH12 In built-in 3D interference check function, when tool holder 2 and object 1 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#4 TDIST13 In built-in 3D interference check function, when tool 3 and object 1 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#5 TDISH13 In built-in 3D interference check function, when tool holder 3 and object 1 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#6 TDIST14 In built-in 3D interference check function, when tool 4 and object 1 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#7 TDISH14 In built-in 3D interference check function, when tool holder 4 and object 1 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

	#7	#6	#5	#4	#3	#2	#1	#0
10967	TDISH24	TDIST24	TDISH23	TDIST23	TDISH22	TDIST22	TDISH21	TDIST21

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIST21 In built-in 3D interference check function, when tool 1 and object 2 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#1 TDISH21 In built-in 3D interference check function, when tool holder 1 and object 2 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#2 TDIST22 In built-in 3D interference check function, when tool 2 and object 2 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#3 TDISH22 In built-in 3D interference check function, when tool holder 2 and object 2 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#4 TDIST23 In built-in 3D interference check function, when tool 3 and object 2 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#5 TDISH23 In built-in 3D interference check function, when tool holder 3 and object 2 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#6 TDIST24 In built-in 3D interference check function, when tool 4 and object 2 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#7 TDISH24 In built-in 3D interference check function, when tool holder 4 and object 2 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

	#7	#6	#5	#4	#3	#2	#1	#0
10968	TDISH34	TDIST34	TDISH33	TDIST33	TDISH32	TDIST32	TDISH31	TDIST31

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIST31 In built-in 3D interference check function, when tool 1 and object 3 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#1 TDISH31 In built-in 3D interference check function, when tool holder 1 and object 3 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#2 TDIST32 In built-in 3D interference check function, when tool 2 and object 3 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#3 TDISH32 In built-in 3D interference check function, when tool holder 2 and object 3 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#4 TDIST33 In built-in 3D interference check function, when tool 3 and object 3 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#5 TDISH33 In built-in 3D interference check function, when tool holder 3 and object 3 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#6 TDIST34 In built-in 3D interference check function, when tool 4 and object 3 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#7 TDISH34 In built-in 3D interference check function, when tool holder 4 and object 3 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

	#7	#6	#5	#4	#3	#2	#1	#0
10969	TDISH44	TDIST44	TDISH43	TDIST43	TDISH42	TDIST42	TDISH41	TDIST41

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIST41 In built-in 3D interference check function, when tool 1 and object 4 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#1 TDISH41 In built-in 3D interference check function, when tool holder 1 and object 4 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#2 TDIST42 In built-in 3D interference check function, when tool 2 and object 4 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#3 TDISH42 In built-in 3D interference check function, when tool holder 2 and object 4 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#4 TDIST43 In built-in 3D interference check function, when tool 3 and object 4 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#5 TDISH43 In built-in 3D interference check function, when tool holder 3 and object 4 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#6 TDIST44 In built-in 3D interference check function, when tool 4 and object 4 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#7 TDISH44 In built-in 3D interference check function, when tool holder 4 and object 4 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

	#7	#6	#5	#4	#3	#2	#1	#0
10970	TDISH54	TDIST54	TDISH53	TDIST53	TDISH52	TDIST52	TDISH51	TDIST51

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIST51 In built-in 3D interference check function, when tool 1 and object 5 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#1 TDISH51 In built-in 3D interference check function, when tool holder 1 and object 5 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#2 TDIST52 In built-in 3D interference check function, when tool 2 and object 5 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#3 TDISH52 In built-in 3D interference check function, when tool holder 2 and object 5 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#4 TDIST53 In built-in 3D interference check function, when tool 3 and object 5 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#5 TDISH53 In built-in 3D interference check function, when tool holder 3 and object 5 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#6 TDIST54 In built-in 3D interference check function, when tool 4 and object 5 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#7 TDISH54 In built-in 3D interference check function, when tool holder 4 and object 5 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

	#7	#6	#5	#4	#3	#2	#1	#0
10971	TDISH64	TDIST64	TDISH63	TDIST63	TDISH62	TDIST62	TDISH61	TDIST61

[Input type] Parameter input
[Data type] Bit

> ⚠ **CAUTION**
> This parameter is not updated until the power supply is turned off once or built-in 3D interference check setting change signal TDICHG<G519.4> is set to “1”.

#0 TDIST61 In built-in 3D interference check function, when tool 1 and object 6 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#1 TDISH61 In built-in 3D interference check function, when tool holder 1 and object 6 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#2 TDIST62 In built-in 3D interference check function, when tool 2 and object 6 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#3 TDISH62 In built-in 3D interference check function, when tool holder 2 and object 6 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#4 TDIST63 In built-in 3D interference check function, when tool 3 and object 6 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#5 TDISH63 In built-in 3D interference check function, when tool holder 3 and object 6 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#6 TDIST64 In built-in 3D interference check function, when tool 4 and object 6 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

#7 TDISH64 In built-in 3D interference check function, when tool holder 4 and object 6 interfere,
0: OT alarm occurs.
1: OT alarm does not occur, and interference is notified by the signal.

	#7	#6	#5	#4	#3	#2	#1	#0
11005								SIC

[Input type] Parameter input
[Data type] Bit

#0 SIC Spindle indexing is:
0: Performed based on absolute coordinates.
1: Performed based on machine coordinates.

11090	Path number with which the rotation of each spindle is specified

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 10
When a path is specified for spindle commands, this parameter sets a path number with which the rotation of a spindle can be specified.
0: Spindle commands can be issued from all paths.
1 to 10: Spindle commands can be issued from a set path.

NOTE
1 This parameter is valid when SPSP<Gn536.7> is set to 1.
2 If the setting is illegal, an alarm PS5305, “ILLEGAL SPINDLE NUMBER” is issued when a spindle command is issued from any one of the paths.
3 This setting does not apply to spindle commands using the spindle select signals (SWS1 to SWS4<Gn027.0 to Gn027.2, Gn026.3>).

11100	M code for Path Table Operation

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 1 to 99999999
M code for starting Path Table Operation is specified. The value should be out of the waiting M code (Less than Parameter No.8110 or larger than parameter No.8111.)

11101	Tolerance between actual position and command value of Path Table start block

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch, degree (input unit)
[Valid data range] 0 or 9-digit of least input increment. In case of IS-B, the valid value is 0 or 0.001 to +999999.999.
At the start of the Path Table Operation, the difference between the actual axis coordinate and the axis command value is checked. If the difference exceeds the parameter, the alarm is generated. If 0 is set in the parameter, no check is carried out.

11102	Tolerance between actual spindle speed and command value of Path Table start block

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0, 1 to +99999999

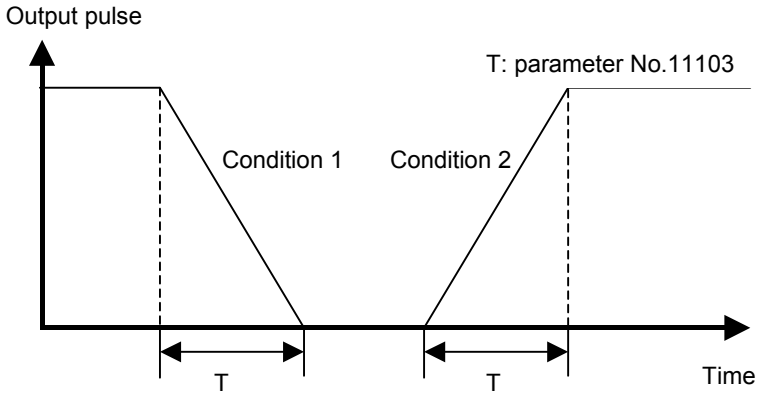
At the start of the Path Table Operation, the difference between the actual spindle speed and commanded spindle speed at the spindle command table is checked. If the difference exceeds the parameter, the alarm is generated. If 0 is set in the parameter, no check is carried out.

NOTE
In case of the slave spindle of synchronized spindle control or constant surface speed control, no check is carried out.

11103	Time constant of Path Table Operation in deceleration stop or restart

[Input type] Parameter input
[Data type] Word path
[Unit of data] msec
[Valid data range] 0 to 32767

The time constant for feed hold, reset or alarm status in Path Table Operation, is set (Condition 1). The time constant for restart in Path Table Operation, is set(Condition 2). The parameter is set in path(head) 1. The parameter is common to all path(head).



NOTE
When setting the larger value in this parameter, the deceleration time expands and the distance to stop becomes longer. This parameter is not effective for the emergency stop or servo alarm stop.

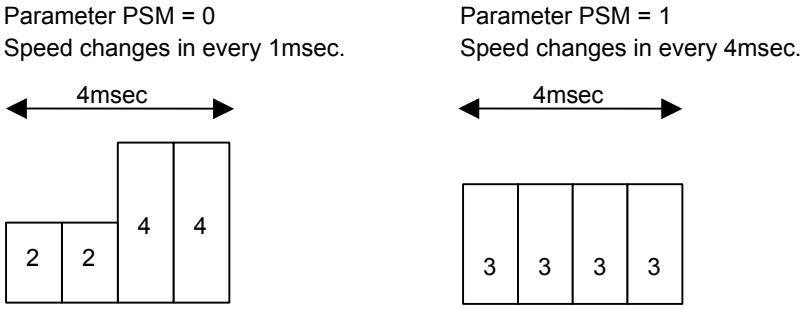
	#7	#6	#5	#4	#3	#2	#1	#0
11104							PSM	PCA

[Input type] Parameter input
[Data type] Bit

 # 0 PCA When the Path Table operation mode, the position command and the standard coordinate command is:
0: input unit of the value of the machine coordinate system.
1: input unit of the value of the workpiece coordinate system.

#1 PSM In Path Table Operation, smoothing of axis movement commands is:
0: not executed.
1: executed.

When bit 1 (PSM) of parameter No.11104 is set to 1, the reductive effect of torque command by acceleration/deceleration after interpolation becomes larger because the timing of changing speed by axis movement commands is in every 4msec. If bit 1 (PSM) of parameter No.11104 is set, set the time constant of acceleration/deceleration after interpolation(parameter No.11110) too. As the time constant of acceleration/deceleration after interpolation is larger, the reductive effect of torque command by bit 1 (PSM) of parameter No.11104 becomes larger.



11108	The amount of a change in the override every 4msec for Path Table Operation

[Input type] Parameter input
[Data type] Byte
[Unit of data] %
[Valid data range] 0, 1 to 100

The amount of a change in the Path Table override every 4msec. When this parameter is set to 1, it takes 400msec until the actual override is 0% if the Path Table override signal G0520 is changed from 100% to 0%. When this parameter value is 0,this function becomes invalid.

	#7	#6	#5	#4	#3	#2	#1	#0
11109							PBBx	

[Input type] Parameter input
[Data type] Bit axis

#1 PBBx Acceleration/deceleration after interpolation in Path Table Operation:
0: linear acceleration/deceleration is applied.
1: bell-shaped acceleration/deceleration is applied.

11110	Time constant of acceleration/deceleration after cutting feed interpolation in Path Table Operation

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000

Set the time constant of acceleration/deceleration after interpolation in Path Table Operation. The time constant of bell-shaped acceleration/deceleration after cutting feed interpolation or linear acceleration/deceleration after cutting feed interpolation for each axis is set. Acceleration/deceleration type is selected with bit 1 (PBBx) of parameter No.11109.

11113	Action when feed hold is detected during the table of spindle position reference being executed

[Input type] Parameter input
[Data type] Byte

[Valid data range] 0, 1

When feed hold is detected during the table of spindle position reference being executed:

0: Alarm PS0452, "ILLEGAL PATH TABLE OPERATION" (detail alarm No.74) is issued, and Path Table Operation of all paths is stopped. The spindle is not stopped.

1: The feed hold of Path Table Operation in all paths is invalid. The spindle is not stopped. If feed hold signal *SP<Gn008.5> is "0" when all tables of spindle position reference is finished, feed hold becomes valid. The stop by reset and the stop by the alarm are valid. The feed hold is valid for the path which is not in Path Table Operation.

NOTE

Feed hold is as follows.

- Feed hold with feed hold signal *SP<Gn008.5> = "0"
- Feed hold when CNC mode is switched from MEM mode to the manual operation mode as JOG mode.
- Feed hold by the alarm occurred in the other path (bit 1 (IAL) of parameter No.8100 = 0).
  (However, if Path Table Operation is executed in the path, Path Table Operation of all paths is stopped.)

	#7	#6	#5	#4	#3	#2	#1	#0
11200					WSK			RCM

[Input type] Setting input
[Data type] Bit path

#0 RCM When workpiece setting error compensation is performed with a 5-axis machine, tool direction compensation (compensation for a rotation axis) is:

0: Not performed.
1: Performed.

NOTE

In case that tilted working plane indexing and tool center point control are used together, set this parameter to "1".

#3 WSK If, during workpiece setting error compensation, system variables #100151 to #100182 (skip coordinates) are read,

0: Values in the workpiece coordinate system can be read.
1: Values in the workpiece setting coordinate system can be read.

This parameter is also applied to system variables #5061 to #5080 (skip coordinates).

11201	The number of decimal places of rotation direction errors in workpiece setting error compensation

[Input type] Setting input
[Data type] Byte path
[Valid data range] 0 to 8

This parameter sets the number of decimal places of rotation direction errors in workpiece setting error compensation.

Parameter No. 11201	1	2	3	4
Least input increment (deg)	0.1	0.01	0.001	0.0001
Maximum settable value (deg)	±99,999,999.9	±9999,999.99	±999,999.999	±99,999.9999

Parameter No. 11201	5	6	7	8
Least input increment (deg)	0.00001	0.000001	0.0000001	0.00000001
Maximum settable value (deg)	±9,999.99999	±999.999999	±99.9999999	±9.99999999

Note, however, that a value from 1 to 8 can be specified in this parameter.
If a value not within the specifiable range is specified in this parameter, the least input increment of the reference axis is followed.

Unit system of reference axis	IS-A	IS-B	IS-C	IS-D	IS-E
Least input increment (deg)	0.01	0.001	0.0001	0.00001	0.000001
Maximum settable value (deg)	±999,999.99	±999,999.999	±99,999.9999	±9,999.99999	±999.999999

11204 **Angle to decide singular posture (for Workpiece setting error compensation)**

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A)
(When the increment system is IS-B, -999999.999 to +999999.999)
When Tool direction is compensated (bit 0 (RCM) of parameter No. 11200 is set to 1), in the case that Tool center point control is active during Workpiece setting error compensation, 3-dimension coordinates system conversion or tilted working plane indexing, rotary axes are compensated. Then, compensation may be different when the tool is in singular posture.
When the angle between the tool posture and the singular posture is less than this parameter, the compensation is done regarding tool posture as singular posture.

11220 **Minimum distance used for determining a plane when a tilted working plane indexing with three points is specified**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
When a tilted working plane indexing with three points is specified, if the distance (used for determining a place) between a straight line passing two points and the remaining one point is short, the plane is unstable. In this parameter, set the minimum distance used for determining a plane. If the distance is shorter than the value set in this parameter, an alarm PS5457, “G68.2/G68.3 FORMAT ERROR” is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
11221					TLC	3DW	D3R	MTW

[Input type] Parameter input
[Data type] Bit path

#0 MTW Multiple tilted working plane indexing is:
0: Not used.
1: Used.

#1 D3R In the 3-dimensional coordinate system conversion mode, tilted working plane indexing mode, or workpiece setting error compensation mode, rapid traverse in canned cycle for drilling is:
0: Performed in the cutting feed mode.
1: Performed in the rapid traverse mode.

#2 3DW If, in the 3-dimensional coordinate system conversion mode, workpiece coordinate system selection using a G code is specified, the selection:
0: Operates in accordance with conventional specifications. (The workpiece coordinate system difference is reflected in the program coordinate system direction.)
1: Operates in accordance with the same specifications as those of workpiece coordinate system selection (bit 6 (3TW) of parameter No. 1205 = 1) during the tilted working plane indexing. (The workpiece coordinate system difference is reflected in the workpiece coordinate system direction.)

> ⚠ **CAUTION**
> If this parameter is set to 1, only G54 to G59 and G54.1 can be specified. If G52 and G92 are specified, alarm PS5462, "ILLEGAL COMMAND (G68.2/G69)" is issued.
> If G54 to G59 and G54.1 are specified, buffering is suppressed.

#3 TLC During tool length compensation, 3-dimensional coordinate conversion:
0: Cannot be used.
1: Can be used.

	#7	#6	#5	#4	#3	#2	#1	#0
11222	PDM					IMG	CIM	NIM

[Input type] Parameter input
[Data type] Bit path

#0 NIM Automatic conversion of a coordinate system by an inch/metric conversion command (G20 or G21) is:
0: Not performed.
1: Performed.

#1 CIM When an inch/metric conversion command (G20 or G21) is specified, if the workpiece coordinate system is shifted by the shift amount as described below:
0: An alarm PS1298, “ILLEGAL INCH/METRIC CONVERSION” is issued.
1: Clearing is performed.

If bit 0 (NIM) of parameter No. 11222 is set to 1, or if bit 2 (IRF) of parameter No. 14000 is set to 1, this parameter clears the following:
- Manual intervention made when the manual absolute signal is off
- Issuance of a move command with the machine locked
- Movement by handle interrupt
- Operation with a mirror image
- Shifting of a workpiece coordinate system when a local coordinate system or workpiece coordinate system is set up

#2 IMG Inch/metric changeover is:
0: Performed with the G20/G21 (G70/G71).
1: Not performed with the G20/G21 (G70/G71).

> **NOTE**
> If bit 2 of parameter No. 11222 is 1 (inch/metric changeover with G20/G21 is disabled), only bit 2 of parameter No. 0 can be used to perform inch/metric changeover. If bit 2 of parameter No.0 = 0, the metric system is used. If bit 2 of parameter No.0 = 1, the inch system is used.

#7 PDM When switching between diameter and radius specification is made with the function for dynamic switching of diameter/radius specification, coordinates in the machine coordinate system select command (G53) are:
0: Switched between diameter and radius specification.
1: Set according to the setting of bit 3 (DIAx) of parameter No. 1006.

	#7	#6	#5	#4	#3	#2	#1	#0
11240								FAE

[Input type] Parameter input
[Data type] Bit path

#0 FAE During positioning when the AI contour control mode is canceled, the optimum torque acceleration/deceleration is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
11250	OAA			MTO	MCO	BOU	TOU	

[Input type] Parameter input
[Data type] Bit path

#1 TOU When the program restart auxiliary function output function is applied in a lathe system, T codes are:
0: Not output to the MDI program.
1: Output to the MDI program.

> **NOTE**
> In a machining center system, they are output regardless of the parameter setting.

#2 BOU When the program restart auxiliary function output function is applied in a lathe system, B codes (second auxiliary function) are:
0: Not output to the MDI program.
1: Output to the MDI program.

> **NOTE**
> In a machining center system, they are output regardless of the parameter setting.

#3 MCO If, in the program restart auxiliary function output function, multiple MSTB codes are specified in the program to restart (or multiple M codes are specified), the output to the MDI program is as follows:
0: Each code is output to a single block.
1: All specified codes are output to a single block.
In either case, the output is in MSTB order.

#4 MTO In the program restart auxiliary function output function, modal T codes are:
0: Not output to the MDI program.
1: Output to the MDI program.

 # 7 OAA In the program restart output function, the approach to the program restart position for each arbitrary axis is:
0: Not used.
1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
11260	TRC	TFD		TDG		AAI		

[Input type] Parameter input
[Data type] Bit path

#2 AAI When tool center point control(G43.4/G43.5), or cutting point command (G43.8/G43.9) is commanded, AI contour control:
0: does not become active automatically.
1: becomes active automatically.

#4 TDG The distance to go during tool center point control is displayed in:
0: the table coordinate system.
1: the machine coordinate system.

NOTE
Even if TDG = 0, the distance to go in the machine coordinate system is displayed in the following modes:
- 3-dimensional coordinate system conversion
- Tilted working plane indexing command
- Workpiece setting error compensation

#6 TFD The actual cutting feedrate displayed during tool center point control is:
0: Control point feedrate.
1: Tool center point feedrate.

#7 TRC Rapid traverse during tool center point control results in:
0: Tool path where tool center point control is enabled.
1: Tool path where tool center point control is disabled.

NOTE
This parameter is regarded as being set to 0 in the following modes:
1) 3-dimensional coordinate system conversion
2) Tilted working plane indexing
3) Workpiece setting error compensation
4) Cutting point command
If 1 is set in this parameter, manual intervention cannot be performed in rapid traverse during tool center point control. If manual intervention is performed, alarm PS5421," ILLEGAL COMMAND IN G43.4/G43.5" is issued at the cycle start after manual intervention.

11261	The amount of a retract operation in the tool axis direction during tool retract and return

[Input type] Setting input
[Data type] Real axis
[Unit of data] mm, inch, degree (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the amount of a retract operation in the tool axis direction when G10.6 is specified alone during tool retraction and return (during tool center point control and workpiece setting error compensation). The retract operation is performed by using the value set in this parameter in an incremental manner. This data is valid only when bit 2 (RPS) of parameter No. 7040 is set to 1. If 0 is set in this parameter, a retract operation in the tool axis direction is not performed, but a retract operation is performed according to the setting of parameter No. 7041.

11262	Angle decided to result in a special point posture (cutting point command)

[Input type] Parameter input
[Data type] Real path
[Unit of data] degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
If the angle formed by the tool posture and the vertical direction of the cutting plane is equal to or less than the value set in this parameter, the tool posture is regarded as a posture close to a special point. In the event of a posture close to a special point, the "vector pointing to the center point from the specified point" is fixed with the "vector pointing to the center point from the cutting point" immediately before the posture close to a special point.

11266	Active machine configuration set number

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 10
The number of machine configuration sets that can be used in all paths is up to 10.

When the total number of the sets exceeds 10, the set number is assigned in order with small path number by priority.
Example)
In case of setting as the followings, Path1: 6sets, Path2: 8sets, Path3: 4sets.
Actually the set number is assigned as the followings, Path1: 6sets, Pathe2: 4sets, Path3: 0set.
In case of setting all zero for all paths, 10 sets are used for the 1st path.

NOTE
When this parameter is set, the saved machine configuration data is cleared. The backup to external devices is recommended before it is set.

	#7	#6	#5	#4	#3	#2	#1	#0
11268								ADXx

[Input type] Parameter input
[Data type] Bit axis

#0 ADXx In tool center point control, the axis which is specified by this parameter is
0: Not included in commanded feedrate.
1: Included in commanded feedrate.

NOTE
- It is not necessary to set this parameter for the basic three axes. Even if this parameter for the basic three axes is set, it makes no effect.
- In case that expansion of axis move command in tool center point control is used and parameter ADF(No.11269#2)=1, all the non 5-axis machining control axes are included in commanded feedrate regardless of the setting of parameter ADXx.

	#7	#6	#5	#4	#3	#2	#1	#0
11269	MSF				ON1	ADF		

[Input type] Parameter input
[Data type] Bit path

#2 ADF In case that expansion of axis move command in tool center point control is used, this parameter determines whether all the non 5-axis machining control axes are included in commanded feedrate or not.
0: Parameter ADXx(No.11268#0) determines whether each non 5-axis machining control axis is included in commanded feedrate or not.
1: All the non 5-axis machining control axes are included in commanded feedrate.

NOTE
- "Non 5-axis machining control axis" means the axis which is not subject to tool center point control.
- In case that the number of non 5-axis machining control axes are two or more and some of them should be included in commanded feedrate but others should be not, set parameter ADF=0 and set parameter ADXx(No.11268#0) for each axis individually.

#3 ON1 When cutting point command is used with tool compensation memory C of machining center system, the specification method of tool length, radius and corner-R is:
0: Specified individually by two tool offset numbers.
1: Specified collectively by one tool offset numbers.

#7 MSF When G code (G10.8L3) of the switching machine configuration is specified with the option of the machine configuration selecting function
0: Disabled. The alarm PS0010 "IMPROPER G-CODE" is issued.
1: Enabled.

11290	M code preventing buffering 11
11291	M code preventing buffering 12
to	to
11299	M code preventing buffering 20

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 99999999
Set M codes that prevent buffering the following blocks. If processing directed by an M code must be performed by the machine without buffering the following block, specify the M code.
M00, M01, M02, and M30 always prevent buffering even when they are not specified in these parameters.

	#7	#6	#5	#4	#3	#2	#1	#0
11304					ON8		GGD	

[Input type] Parameter input
[Data type] Bit

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#1 GGD The G code guidance screen is:
0: Not displayed.
1: Displayed.

#3 ON8 Program numbers are:
0: Four digits long.
1: Eight digits long.

NOTE
If program numbers are changed from eight digits to four digits, all programs will be automatically deleted from program memory.
If this parameter is changed from 1 to 0 and the power is turned off and back on, the following message appears on the IPL screen. For the Series 30*i*/31*i*/32*i* (with personal computer function with Windows CE) and for the CNC screen display function, the message appears on the IPL screen of NCBOOT32.exe. To delete them, enter 1. Otherwise, enter 0.
PARAMETER NO.11304#3 IS CHANGED.
ALL PROGRAM FILE MUST BE CLEARED.
CLEAR FILE OK ? (NO=0, YES=1)

11305	Maximum number of simultaneously displayed axes

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 2

By setting this parameter, the maximum number of simultaneously displayed axes at the current position can be changed. A value set in this parameter corresponds to the maximum number of simultaneously displayed axis as follows:

Max. number of simultaneously displayed axes	5	10	20
Setting	0	1	2

A value other than 1 and 2 is assumed to be 0.

11307	Display sequence of the coordinates in current position display

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 5

This parameter sets the display sequence of the coordinates of a position displayed on the following screens:
10.4-, 15-, and 19-inch display units
- Total position display screen
- Total position display on each screen

8.4-inch display unit
- Total position display screen

When the maximum number of simultaneously displayed axes is set to 20 (when 2 is set in parameter No. 11305), two sets of coordinates are displayed simultaneously as the current position display on each screen.

When the first set is displayed, switching to the second set can be made by pressing  then pressing the chapter selection soft key being selected.

When the above operation is performed again, the displayed set changes to the first set.
The display sequence of coordinates corresponds to the parameter setting as follows:

Setting \ Display sequence of coordinates	1	2	3	4
0	Relative coordinates	Absolute coordinates	Machine coordinates	Remaining travel distance
1	Relative coordinates	Machine coordinates	Absolute coordinates	Remaining travel distance
2	Relative coordinates	Remaining travel distance	Absolute coordinates	Machine coordinates
3	Absolute coordinates	Machine coordinates	Relative coordinates	Remaining travel distance
4	Absolute coordinates	Remaining travel distance	Relative coordinates	Machine coordinates
5	Machine coordinates	Remaining travel distance	Relative coordinates	Absolute coordinates

If the setting is beyond the valid data range, 0 is assumed.
When the multi-path simultaneous display function is enabled (parameter No. 13131 is set to a nonzero value, and parameter No. 13132 is set to 1 or a greater value), this parameter becomes invalid.

	#7	#6	#5	#4	#3	#2	#1	#0
11308					FPD		COW	

[Input type] parameter input
[Data type] Bit

#1 COW When the file of specified name already exists on memory card,
0: It is not overwritten
Alarm SR1973, “FILE ALREADY EXIST” is generated.
1: It is overwritten.
Because the confirmation message is displayed before overwriting even if COW is 1, overwriting can be canceled.

NOTE
When the overwritten file is read only attribute, it is not possible to overwrite even if bit 1 (COW) of parameter No. 11308 = 1.

#3 FPD On the program screen and program check screen, blocks already executed are:
0: Not displayed.
1: Displayed.

NOTE
This parameter is effective at bit 1 (APD) of parameter No.11350=0.

	#7	#6	#5	#4	#3	#2	#1	#0
11329	GST				BGM			

[Input type] Parameter input

[Data type] Bit path

#3 BGM Coordinates used by the dynamic graphic display function are:
0: Absolute coordinates.
1: Machine coordinates.

#7 GST When drawing cannot be performed for a command with the dynamic graphic display function:
0: The command is ignored, and drawing continues without stopping drawing.
1: Drawing stops.

11343	Blank figure in dynamic graphic display

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 1
This parameter sets the type of a blank figure in dynamic graphic display.

Setting	Figure
0	Cylinder or hollow cylinder (parallel to the Z-axis)
1	Rectangular parallelepiped

11344	Blank reference position in dynamic graphic display

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm,inch (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the reference position of a blank in the dynamic graphic display function by using coordinate values in the workpiece coordinate system.

NOTE
If bit 3 (BGM) of parameter No. 11329 is set to 1, set coordinate values in the machine coordinate system.

11345	Blank dimension I in dynamic graphic display
11346	Blank dimension J in dynamic graphic display
11347	Blank dimension K in dynamic graphic display

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm,inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.000 to +999999.999)
These parameters set the dimensions of a blank in the dynamic graphic display function according to the blank figure as follows:

Blank type	Dimension I	Dimension J	Dimension K
Cylinder	Column diameter	0	Column length

Blank type	Dimension I	Dimension J	Dimension K
Hollow cylinder	Diameter of outer circle of cylinder	Diameter of inner circle of cylinder	Cylinder length
Rectangular prism	Length in X-axis direction	Length in Y-axis direction	Length in Z-axis direction

	#7	#6	#5	#4	#3	#2	#1	#0
11349					WNS		ABC	

[Input type] Parameter input
[Data type] Bit

#1 ABC In animated simulation in the dynamic graphic display function, when a fine boring cycle or back boring cycle, which is a hole machining canned cycle, is performed, the movement for a shift at the hole bottom is:
0: Not drawn.
1: Drawn.

#3 WNS In the dynamic graphic display function, P-CODE workpiece number search is:
0: Disabled.
1: Enabled.

NOTE
A macro executor option, or a macro executor and C Language Executor options are required.

	#7	#6	#5	#4	#3	#2	#1	#0
11350		QLS		9DE		PNE	APD	

[Input type] Parameter input
[Data type] Bit

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#1 APD The display of the program under execution is:
0: A display containing look-ahead blocks.
1: A text display.

#2 PNE Path name expansion display function is:
0: Disabled.
1: Enabled.

NOTE
This parameter is invalid if the number of paths is 1.
It is effective to 10.4-, 15-inch, and 19-inch display units.

#4 9DE On 8.4-inch display unit, the maximum number of axes that can be displayed on a single screen is:
0: 4.
1: 5.

#6 QLS The machining quality level adjustment screen is:
0: Not displayed.
1: Displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
11352					MPC			PNI

[Input type] Parameter input
[Data type] Bit path

#0 PNI The display by the path name enlarged display function is:
0: A normal display.
1: A reverse display.

NOTE
This parameter is effective to 10.4-, 15-, and 19-inch display units.

#3 MPC In this path, the batch making and the batch selection of the multi-path program management function are:
0: effective.
1: invalid.

In the multi-path program management function, the main program in the path that is set parameter MPC=1 is removed from the object of the batch making and batch selection. Please set parameter MPC=1 in the path that should not be machined.

	#7	#6	#5	#4	#3	#2	#1	#0
11353								SEK

[Input type] Parameter input
[Data type] Bit

#0 SEK When the power is turned on, or when the clear state is present, sequence numbers are:
0: Not maintained.
1: Maintained.

NOTE
During a subprogram call, the sequence number of the subprogram is maintained.

	#7	#6	#5	#4	#3	#2	#1	#0
11354				DPC			CRS	

[Input type] Parameter input
[Data type] Bit

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#1 CRS While data transmission is awaited using the DPRNT/BPRNT of the custom macro or macro executor, screen switching is:
0: Not possible.
1: Possible.

#4 DPC In the screen title, program comments corresponding to O-numbers are:
0: Displayed.
1: Not displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
11356								TLD

[Input type] Parameter input
[Data type] Bit

#0 TLD When the protection signal is enabled, the deletion of the tool life management screen is:
0: Disabled.
1: Enabled.

NOTE
The 8-level data protection function cannot be disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
11364					FDR			SFB

[Input type] Parameter input
[Data type] Bit

#0 SFB Folder made by template program function or multi-path program management function
0: should be set to the foreground folder and the background folder by an operator.
1: becomes to the foreground folder and the background folder automatically.

NOTE
When the item of "CAN NOT ENTER MULTIPATH PROG FOLDER" is effective in the operation confirmation function setting screen, this parameter is invalid.

#3 FDR If a program or a folder exists in the target folder when the deletion operation is done specifying the folder,
0: the folder is not deleted.
1: the folder and programs/folders in the target folder are deleted.

	#7	#6	#5	#4	#3	#2	#1	#0
11370						RPD		

[Input type] Parameter input
[Data type] Bit

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#2 RPD During executing the program backward by manual handle retrace, the block displayed at the start of the program is:

0: The block being executed.
1: The block just before the block being executed.

> **NOTE**
> This parameter is effective at bit 1 (APD) of parameter No.11350=0 and bit 3 (FPD) of parameter No.11308=1, or bit 1 (APD) of parameter No.11350=1.

	#7	#6	#5	#4	#3	#2	#1	#0
11403				MMT				

[Input type] Parameter input
[Data type] Bit path

#4 MMT If the tool offset for milling and turning function is used, the compensation values acquired with machining center system measurement functions are regarded as:

0: Tool length compensation values.
1: Tool position compensation values.

If, in a machining center system measurement function,
the X-axis direction of the tool is measured with MMT being set to 0, this results in the Z-axis/length (tool length compensation value).
the X-axis direction of the tool is measured with MMT being set to 1, this results in the tool position compensation value in the X-axis direction.

11411	Number of the workpiece coordinate system used as the reference for workpiece setting error amount No. 01
11412	Number of the workpiece coordinate system used as the reference for workpiece setting error amount No. 02
11413	Number of the workpiece coordinate system used as the reference for workpiece setting error amount No. 03
11414	Number of the workpiece coordinate system used as the reference for workpiece setting error amount No. 04
11415	Number of the workpiece coordinate system used as the reference for workpiece setting error amount No. 05
11416	Number of the workpiece coordinate system used as the reference for workpiece setting error amount No. 06
11417	Number of the workpiece coordinate system used as the reference for workpiece setting error amount No. 07

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 1300

These parameters set the numbers of the workpiece coordinate systems used as the reference for the respective workpiece setting errors.
For G54 to G59, set 54 to 59. For G54.1P1 to G54.1P300, set 1001 to 1300.
If 0 is set in one of these parameters, the workpiece setting error corresponding to that parameter cannot be used in multiple workpiece coordinate systems.

11419	The interval of the tool offset number when cutting point command is used with tool compensation memory A and B

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to [(the maximum number of tool compensation values)-1]/2 (omit the figures after the decimal fractions)

When cutting point command is used with tool compensation memory A and B of machining center system, the tool offset number of each compensation value is as follows.
- Tool offset number of tool length compensation value
  H code
- Tool offset number of tool radius compensation value
  H code + (parameter No. 11419)
- Tool offset number of corner-R compensation value
  H code + 2 * (parameter No. 11419)

When parameter No. 11419 is 0, the interval of the offset number is 1.
If parameter No. 11419 is smaller than 0, alarm PS5464, "ILLEGAL COMMAND IN G43.8/G43.9", occurs when cutting point command starts.
If parameter No. 11419 is larger than [(the maximum number of tool compensation values)-1]/2, alarm PS0030, "ILLEGAL OFFSET NUMBER", occurs when cutting point command starts.

	#7	#6	#5	#4	#3	#2	#1	#0
11420								RAU

[Input type] Parameter input
[Data type] Bit path

#0 RAU Optimum torque acceleration/deceleration function for rigid tapping is
0: Disabled.
1: Enabled.

11421	Maximum acceleration of the optimum acceleration/deceleration for rigid tapping (gear 1)
11422	Maximum acceleration of the optimum acceleration/deceleration for rigid tapping (gear 2)
11423	Maximum acceleration of the optimum acceleration/deceleration for rigid tapping (gear 3)
11424	Maximum acceleration of the optimum acceleration/deceleration for rigid tapping (gear 4)

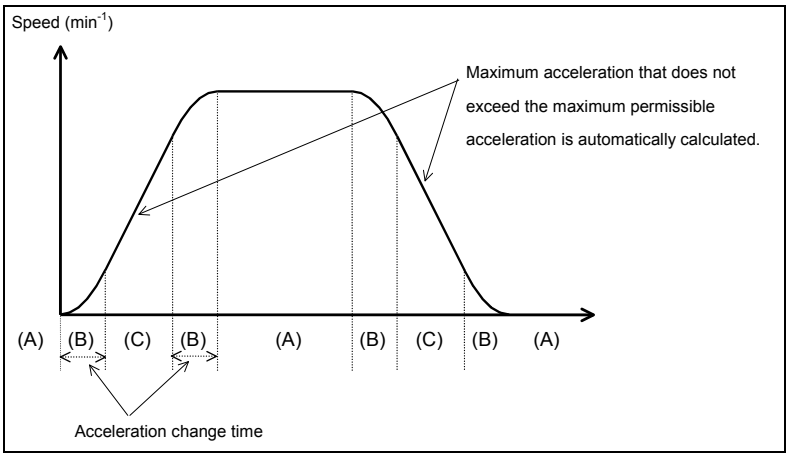
[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] rev/sec^2
[Valid data range] 0 to 10000.0
These parameters set maximum accelerations.

11425	Acceleration change time of bell-shaped acceleration/deceleration in optimum acceleration/deceleration for rigid tapping (gear 1)
11426	Acceleration change time of bell-shaped acceleration/deceleration in optimum acceleration/deceleration for rigid tapping (gear 2)
11427	Acceleration change time of bell-shaped acceleration/deceleration in optimum acceleration/deceleration for rigid tapping (gear 3)

11428	Acceleration change time of bell-shaped acceleration/deceleration in optimum acceleration/deceleration for rigid tapping (gear 4)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] msec
[Valid data range] 0 to 200

These parameters set the acceleration change time of bell-shaped acceleration/deceleration in optimum acceleration/deceleration for rigid tapping (time taken for the change from the constant speed state (A) to the acceleration state (C) with the acceleration calculated from the optimum acceleration/deceleration for rigid tapping, i.e., the time indicated by (B) in the figure below).



11429	Spindle speed at P1 in optimum acceleration/deceleration for rigid taping (gear 1)
11430	Spindle speed at P2 in optimum acceleration/deceleration for rigid taping (gear 1)
11431	Spindle speed at P3 in optimum acceleration/deceleration for rigid taping (gear 1)
11432	Spindle speed at P1 in optimum acceleration/deceleration for rigid taping (gear 2)
11433	Spindle speed at P2 in optimum acceleration/deceleration for rigid taping (gear 2)
11434	Spindle speed at P3 in optimum acceleration/deceleration for rigid taping (gear 2)
11435	Spindle speed at P1 in optimum acceleration/deceleration for rigid taping (gear 3)
11436	Spindle speed at P2 in optimum acceleration/deceleration for rigid taping (gear 3)
11437	Spindle speed at P3 in optimum acceleration/deceleration for rigid taping (gear 3)
11438	Spindle speed at P1 in optimum acceleration/deceleration for rigid taping (gear 4)
11439	Spindle speed at P2 in optimum acceleration/deceleration for rigid taping (gear 4)
11440	Spindle speed at P3 in optimum acceleration/deceleration for rigid taping (gear 4)

[Input type] Parameter input
[Data type] Byte spindle
[Unit of data] %
[Valid data range] 0 to 100

These parameters set the spindle speeds at P1 to P3 of acceleration points P0 to P4 as ratios to the maximum spindle speed (parameters Nos. 5241 to 5244). The spindle speed at P0 is 0, while the spindle speed at P4 is the maximum spindle speed. Any acceleration setting points where 0 is set will be skipped.

11441	Permissible acceleration at P0 in optimum acceleration/deceleration for rigid tapping (gear 1)
11442	Permissible acceleration at P1 in optimum acceleration/deceleration for rigid tapping (gear 1)
11443	Permissible acceleration at P2 in optimum acceleration/deceleration for rigid tapping (gear 1)
11444	Permissible acceleration at P3 in optimum acceleration/deceleration for rigid tapping (gear 1)
11445	Permissible acceleration at P4 in optimum acceleration/deceleration for rigid tapping (gear 1)
11446	Permissible acceleration at P0 in optimum acceleration/deceleration for rigid tapping (gear 2)
11447	Permissible acceleration at P1 in optimum acceleration/deceleration for rigid tapping (gear 2)
11448	Permissible acceleration at P2 in optimum acceleration/deceleration for rigid tapping (gear 2)
11449	Permissible acceleration at P3 in optimum acceleration/deceleration for rigid tapping (gear 2)
11450	Permissible acceleration at P4 in optimum acceleration/deceleration for rigid tapping (gear 2)
11451	Permissible acceleration at P0 in optimum acceleration/deceleration for rigid tapping (gear 3)
11452	Permissible acceleration at P1 in optimum acceleration/deceleration for rigid tapping (gear 3)
11453	Permissible acceleration at P2 in optimum acceleration/deceleration for rigid tapping (gear 3)
11454	Permissible acceleration at P3 in optimum acceleration/deceleration for rigid tapping (gear 3)
11455	Permissible acceleration at P4 in optimum acceleration/deceleration for rigid tapping (gear 3)
11456	Permissible acceleration at P0 in optimum acceleration/deceleration for rigid tapping (gear 4)
11457	Permissible acceleration at P1 in optimum acceleration/deceleration for rigid tapping (gear 4)
11458	Permissible acceleration at P2 in optimum acceleration/deceleration for rigid tapping (gear 4)
11459	Permissible acceleration at P3 in optimum acceleration/deceleration for rigid tapping (gear 4)
11460	Permissible acceleration at P4 in optimum acceleration/deceleration for rigid tapping (gear 4)

[Input type] Parameter input
[Data type] Byte spindle
[Unit of data] %
[Valid data range] 0 to 100

These parameters set the permissible accelerations at acceleration setting points P0 to P4 as ratios to the maximum acceleration (parameters Nos. 11421 to 11424). At any acceleration setting points where 0 is set, 100% is assumed.

11461	Permissible deceleration at P0 in optimum acceleration/deceleration for rigid tapping (gear 1)
11462	Permissible deceleration at P1 in optimum acceleration/deceleration for rigid tapping (gear 1)
11463	Permissible deceleration at P2 in optimum acceleration/deceleration for rigid tapping (gear 1)
11464	Permissible deceleration at P3 in optimum acceleration/deceleration for rigid tapping (gear 1)
11465	Permissible deceleration at P4 in optimum acceleration/deceleration for rigid tapping (gear 1)
11466	Permissible deceleration at P0 in optimum acceleration/deceleration for rigid tapping (gear 2)

11467	Permissible deceleration at P1 in optimum acceleration/deceleration for rigid tapping (gear 2)
11468	Permissible deceleration at P2 in optimum acceleration/deceleration for rigid tapping (gear 2)
11469	Permissible deceleration at P3 in optimum acceleration/deceleration for rigid tapping (gear 2)
11470	Permissible deceleration at P4 in optimum acceleration/deceleration for rigid tapping (gear 2)
11471	Permissible deceleration at P0 in optimum acceleration/deceleration for rigid tapping (gear 3)
11472	Permissible deceleration at P1 in optimum acceleration/deceleration for rigid tapping (gear 3)
11473	Permissible deceleration at P2 in optimum acceleration/deceleration for rigid tapping (gear 3)
11474	Permissible deceleration at P3 in optimum acceleration/deceleration for rigid tapping (gear 3)
11475	Permissible deceleration at P4 in optimum acceleration/deceleration for rigid tapping (gear 3)
11476	Permissible deceleration at P0 in optimum acceleration/deceleration for rigid tapping (gear 4)
11477	Permissible deceleration at P1 in optimum acceleration/deceleration for rigid tapping (gear 4)
11478	Permissible deceleration at P2 in optimum acceleration/deceleration for rigid tapping (gear 4)
11479	Permissible deceleration at P3 in optimum acceleration/deceleration for rigid tapping (gear 4)
11480	Permissible deceleration at P4 in optimum acceleration/deceleration for rigid tapping (gear 4)

[Input type] Parameter input
[Data type] Byte spindle
[Unit of data] %
[Valid data range] 0 to 100

These parameters set the permissible decelerations at acceleration setting points P0 to P4 as ratios to the maximum acceleration (parameters Nos. 11421 to 11424). At any acceleration setting points where 0 is set, 100% is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
11501						MSC		

[Input type] Parameter input
[Data type] Bit path

#2 MSC The machine lock shift value check in automatic operation is :
0: Not applied.
1: Applied.

NOTE
1 This function is invalid for the axis in the machine lock state.
2 The followings can be commanded with the machine lock shift value remained.
- Automatic reference position return (The middle point is not specified)
- Workpiece coordinate system preset
- Local coordinate system setting
- Machine coordinate system setting
- Coordinate system setting
- Workpiece coordinate system setting (Axis movement is not specified)

NOTE
3 In other automatic operation, if there is no movement in the machine coordinate system, the alarm is not issued.
4 This function is invalid for the dummy axis. (Parameter KSV(No.11802#4)=1 or DMY(No.2009#0)=1)

	#7	#6	#5	#4	#3	#2	#1	#0
11502	IPW	CTC				WPP		

[Input type] Parameter input
[Data type] Bit

#2 WPP Programmable parameter input (G10)-based parameter re-setting that requires power-off is:
0: Disabled.
1: Enabled.

NOTE
Setting bit 2 (WPP) of parameter No. 11502 to “1” enables programmable parameter input (G10)-based parameter re-setting that requires power-off even when "PARAMETER WRITE " is disabled.

#6 CTC During axis moving, the time constant of rapid traverse linear acceleration/deceleration for each axis (parameter No. 1620) is:
0: Write-disabled.
1: Write-enabled.

#7 IPW The advanced preview feed-forward coefficient (parameter No. 2092) and bit 0 (SMR) of parameter No. 8162 for specifying whether to apply a mirror image during synchronization control are:
0: Write-disabled during axis moving.
1: Write-enabled if the corresponding axis is stopped.

	#7	#6	#5	#4	#3	#2	#1	#0
11505								ISU

[Input type] Setting input
[Data type] Bit

#0 ISU When an USB memory card is selected as an I/O device, data input/output is performed using
0: ASCII codes.
1: ISO codes.

⚠ WARNING
1 Unless data is input using ASCII codes, set this parameter to 1 to input or output data using ISO codes.
2 Data input/output with ASCII codes is dangerous because parity information is not included and a data error during the data input/output is not detected.

	#7	#6	#5	#4	#3	#2	#1	#0
11506								PCU

[Input type] Parameter input
[Data type] Bit

#0 PCU If there is a USB memory interface on the CNC side, the USB memory interface used when the CNC screen display function is started via the HSSB interface is on the:
0: CNC side.
1: PC side.

NOTE
1 With the PC function, a USB memory interface on the PC side is used regardless of the setting of this parameter if there is no USB memory interface on the CNC main unit side.
2 This parameter is valid only when the CNC screen display function is active via the HSSB interface.
When setting 1 to this parameter, the CNC screen display function must always be executed via the HSSB interface.
3 When using the CNC screen display function via the Ethernet interface, set 0.
4 When this parameter is set, the power must be turned off before operation is continued.

11604	Number of look-ahead blocks for high-speed processing

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 1000
Set the number of look-ahead blocks for each path used in high-speed processing.

NOTE
1 When this parameter is set, the power must be turned off before operation is continued.
2 This parameter can set up to 600 look-ahead blocks for high-speed processing. Extending the number to up to 1000 blocks (optional) enables up to 1000 look-ahead blocks to be set.
3 If the parameter is 0 for all paths, it is assumed that the maximum number of blocks is set for the first path.
4 High-speed processing is usable for up to two paths with up to 12 axes.

	#7	#6	#5	#4	#3	#2	#1	#0
11630						TFR	MDE	FRD

[Input type] Parameter input
[Data type] Bit path

#0 FRD The minimum command unit of the rotation angles of coordinate rotation and 3-dimensional coordinate system conversion is:
0: 0.001 degree.
1: 0.00001 degree. (1/100,000)

#1 MDE In the MDI mode, the external device subprogram call (M198) is:
0: Disabled.
1: Enabled.

NOTE
If instructing M198 at the parameter MDE(No.11630#1)=0, the alarm PS1081"EXT DEVICE SUB PROGRAM CALL MODE ERROR" is generated.

#2 TFR The minimum command unit of the rotation angles of the tilted working plane indexing is:
0: 0.001 degree.
1: 0.00001 degree.

11647	The local variable number corresponding to the axis address

[Input type] Parameter input
[Data type] Byte axis
[Unit of data] No unit
[Valid data range] 0 to 33
The address as an argument of the macro call is allocated to local variable number.
By this parameter, the address of axis name expansion can be the argument of the macro call.

	#7	#6	#5	#4	#3	#2	#1	#0
11651	DCO							

[Input type] Parameter input
[Data type] Bit path

#7 DCO During dry run, the cutting time is:
0: Not counted.
1: Counted.

11681	Smoothing level currently selected when nano smoothing is used

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 10
This parameter sets the smoothing level currently selected when nano smoothing or nano smoothing 2 is used.

11682	Tolerance when nano smoothing is used (smoothing level 1)
11683	Tolerance when nano smoothing is used (smoothing level 10)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, degree (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
Each of these parameters sets a tolerance value when nano smoothing is used.

It is necessary to set the value of both level1 and level10.

11684	Tolerance of rotary axes when nano smoothing 2 is used (smoothing level 1)

11685	Tolerance of rotary axes when nano smoothing 2 is used (smoothing level 10)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, degree (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
Each of these parameters sets tolerances of rotary axes when nano smoothing 2 is used.
It is necessary to set the value of both level1 and level10.

11686	Standard value of smoothing level when nano smoothing is used

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 10
Set standard value of smoothing level when nano smoothing or nano smoothing 2 is used.
When the power is turned on or the system is reset, the smoothing level
0 : keeps its value.
1 to 10 : becomes the level set to this parameter.

11687	Standard value of precision level when AI contour control is used

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 10
Set standard value of precision level when AI contour control is used.
When the power is turned on or the system is reset, the precision level
0 : keeps its value.
1 to 10 : becomes the level set to this parameter.

11750	Tolerance for assuming that the workpiece setting error Δa is 0

11751	Tolerance for assuming that the workpiece setting error Δb is 0

11752	Tolerance for assuming that the workpiece setting error Δc is 0

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Least input increment of the error in the rotation direction in workpiece setting error compensation (see the explanation of parameter No. 11201)
[Unit of data] 0 to 999999999
If the absolute value of the error in the rotation axis in workpiece setting error compensation is equal to or less than the setting of the corresponding one of these parameters, the error in the rotation direction is regarded as 0.
If the setting is negative, its absolute value is used.
The least input increment of the error in the rotation direction depends on the setting of parameter No. 11201.
For a machine with a table rotation axis, these parameters are effective to the errors Δa, Δb, and Δc in the rotation direction, respectively, when the table rotation axis position in the workpiece coordinate system is 0.

11753	Upper limit on the workpiece setting error Δa
11754	Upper limit on the workpiece setting error Δb
11755	Upper limit on the workpiece setting error Δc

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A)
(When the increment system is IS-B, -999999.999 to +999999.999)
If a value other than 0 is set in one of these parameters, and the corresponding error in the rotation axis in workpiece setting error compensation is equal to or greater than the setting of that parameter, alarm PS0517, "SETTING ERROR AMOUNT IS OUT OF RANGE" is issued when workpiece setting error compensation is started.
The least input increment of these parameters adheres to the reference axis regardless of the setting of parameter No. 11201.
For a machine with a table rotation axis, these parameters are effective to the errors Δa, Δb, and Δc in the rotation direction, respectively, when the table rotation axis position in the workpiece coordinate system is 0.

11756	Lower limit on the workpiece setting error Δa
11757	Lower limit on the workpiece setting error Δb
11758	Lower limit on the workpiece setting error Δc

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A)
(When the increment system is IS-B, -999999.999 to +999999.999)
If a value other than 0 is set in one of these parameters, and the corresponding error in the rotation axis in workpiece setting error compensation is equal to or less than the setting of that parameter, alarm PS0517, "SETTING ERROR AMOUNT IS OUT OF RANGE" is issued when workpiece setting error compensation is started.
The least input increment of these parameters adheres to the reference axis regardless of the setting of parameter No. 11201.
For a machine with a table rotation axis, these parameters are effective to the errors Δa, Δb, and Δc in the rotation direction, respectively, when the table rotation axis position in the workpiece coordinate system is 0.

	#7	#6	#5	#4	#3	#2	#1	#0
11775								TP2

[Input type] Setting input
[Data type] Bit path

#0 TP2 At the TCP starting block(G43.4, G43.5), when the address P is omitted, in accordance with the parameter TPC(No.19604#0),
0: Tool center point control or Tool posture control (G43.4P1, G43.5P1) is effective.
1: Tool center point control or Smooth control (G43.4P3, G43.5P3) is effective.

11776 **Tolerance of Tool center point path for High-speed Smooth TCP (G43.4P3, G43.5P3)**

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
In Smooth control (G43.4P3, G43.5P3), paths of Tool center point are paths within this parameter from the commanded linear paths of Tool center points.
When the data 0 is set in this parameter, paths are calculated assuming that the data 0.02 is set in this parameter.

11777 **Tolerance of Tool posture control point path for High-speed Smooth TCP (G43.4P3, G43.5P3)**

[Input type] Setting input
[Data type] Real path
[Unit of data] degree (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 to 90
In High-speed Smooth TCP (G43.4P3, G43.5P3), angles between surfaces of High-speed Smooth TCP and planes of traditional Tool posture control(planes between starting tool posture and ending tool posture of a block) are within this parameter.
When the data 0 is set in this parameter, paths are calculated assuming that the data 0.02 is set in this parameter.

11778 **Longest block length to enable Smoothing of High-speed Smooth TCP (G43.4P3, G43.5P3)**

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
The longest block length (Maximum block length) of Tool center points to enable Smoothing of High-speed Smooth TCP (G43.4P3, G43.5P3) is set.
At blocks whose lengths are longer than this parameter, Smoothing of High-speed Smooth TCP is temporarily canceled.
When the data 0 is set in this parameter, the decision to cancel Smoothing with blocks lengths is not done.

11779 **Maximum angle between blocks to enable Smoothing of High-speed Smooth TCP (G43.4P3, G43.5P3)**

[Input type] Setting input
[Data type] Real path
[Unit of data] degree (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 to 180
Maximum angle between blocks of Tool center points to enable Smoothing of Smooth control (G43.4P3, G43.5P3) is set. When angles between blocks are larger, Smoothing of High-speed Smooth TCP (G43.4P3, G43.5P3) is temporarily canceled.
When the data 0 is set in this parameter, the decision to cancel Smoothing with angles between blocks is done assuming that the data 90 is set in this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
11802				KSVx				

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#4 KSVx Servo axis is:
0: Enabled.
1: Disabled.

NOTE
1 This setting is effective regardless of the value of parameter No. 1023.
2 If this setting is made for the axis subject to Cs axis contour/spindle positioning, Cs axis contour/spindle positioning will be disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
11850	IFH							CMI

[Input type] Parameter input
[Data type] Bit path

#0 CMI If, in PMC axis control, a rapid traverse rate is specified with the axis control block data signal, with bit 0 (RPD) of parameter No. 8002 being set to 1, the rapid traverse rate is:
0: Always treated as being in millimeters.
1: Dependent on the setting of bit 0 (INM) of parameter No. 1001.

#7 IFH When bit 2 (OVE) of parameter No. 8001 is set to 1 in PMC axis control, the 1% rapid traverse override signals *EROVs are:
0: On a path-by-path basis. (The first groups in the individual paths (first, fifth, ninth, ..., 33rd, and 37th groups) are used.)
1: On a group-by-group basis.

12310	States of the manual handle feed axis selection signals when tool axis direction handle feed/interrupt and table-based vertical direction handle feed/interrupt are performed

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 24

This parameter sets the states of the manual handle feed axis selection signal (HS1A to HS1E <Gn018.0 to Gn018.3, Gn411.0> for the first manual handle) or the manual handle interrupt axis selection signal (HS1IA to HS1IE <Gn041.0 to Gn041.3, Gn411.4> for the first manual handle) to perform tool axis direction handle feed/interrupt and table-based vertical direction handle feed/interrupt.
The handle for which the signal states are set is determined by parameter No. 12323.

<Table of correspondence with the manual handle feed axis selection signals>
If parameter No. 12323 is set to 1, the states of the manual handle feed axis selection signals or manual handle interrupt axis selection signals for the first manual handle in the 3-dimensional manual feed (handle feed) mode and corresponding parameter settings are listed in the table below. When the first manual handle pulse generator is turned after setting the signals corresponding to the value set in the parameter, operation is performed in the specified mode.

HS1E (HS1IE)	HS1D (HS1ID)	HS1C (HS1IC)	HS1B (HS1IB)	HS1A (HS1IA)	Parameter (No. 12310)
0	0	0	0	1	1
0	0	0	1	0	2
0	0	0	1	1	3
0	0	1	0	0	4
0	0	1	0	1	5
0	0	1	1	0	6
0	0	1	1	1	7
0	1	0	0	0	8
0	1	0	0	1	9
0	1	0	1	0	10
0	1	0	1	1	11
0	1	1	0	0	12
0	1	1	0	1	13
0	1	1	1	0	14
0	1	1	1	1	15
1	0	0	0	0	16
1	0	0	0	1	17
1	0	0	1	0	18
1	0	0	1	1	19
1	0	1	0	0	20
1	0	1	0	1	21
1	0	1	1	0	22
1	0	1	1	1	23
1	1	0	0	0	24

If parameter No. 12323 is set to 2 to 5, replace 1 in HS1A to HS1E and HS1IA to HS1IE above with 2 to 5.

12311	States of the manual handle feed axis selection signals when a movement is made in the first axis direction in tool axis normal direction handle feed/interrupt and table-based horizontal direction handle feed/interrupt

[Input type] Parameter input

[Data type] Byte path
[Valid data range] 1 to 24

This parameter sets the states of the manual handle feed axis selection signals (HS1A to HS1E <Gn018.0 to Gn018.3, Gn411.0> for the first manual handle) or the manual handle interrupt axis selection signal (HS1IA to HS1IE <Gn041.0 to Gn041.3, Gn411.4> for the first manual handle) when a movement is made in the first axis direction. (For settings, see "Table of correspondence with the manual handle feed axis selection signals" in the description of parameter No. 12310.)
The handle for which the signal states are set is determined by parameter No. 12323.

The table below indicates the relationships of tool axis directions, first axis directions, and second axis directions.

Parameter No. 19697	Tool axis directions	First axis directions	Second axis directions
1	X	Y	Z
2	Y	Z	X
3	Z	X	Y

Note, however, that the table above indicates the directions applicable when the angles of all rotation axes are set to 0.
In tool axis direction/tool axis normal direction feed (not table-based), the directions indicated above assume that 0 is set in parameter No. 19698 and No. 19699. When a rotation axis has made a turn or a nonzero value is set in these parameters in tool axis direction/tool axis normal direction feed, the relevant directions are inclined accordingly.

12312 **States of the manual handle feed axis selection signals when a movement is made in the second axis direction in tool axis normal direction handle feed/interrupt and table-based horizontal direction handle feed/interrupt**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 24

This parameter sets the states of the manual handle feed axis selection signals (HS1A to HS1E <Gn018.0 to Gn018.3, Gn411.0> for the first manual handle) or the manual handle interrupt axis selection signals (HS1IA to HS1IE <Gn041.0 to Gn041.3, Gn411.4> for the first manual handle) when a movement is made in the second axis direction. (For settings, see "Table of correspondence with the manual handle feed axis selection signals" in the description of parameter No. 12310.)
The handle for which the signal states are set is determined by parameter No. 12323.

12313 **States of the manual handle feed axis selection signals when the first rotation axis is turned in tool tip center rotation handle feed/interrupt**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 24

This parameter sets the states of the manual handle feed axis selection signals (HS1A to HS1E <Gn018.0 to Gn018.3, Gn411.0> for the first manual handle) or the manual handle interrupt axis selection signals (HS1IA to HS1IE <Gn041.0 to Gn041.3, Gn411.4> for the first manual handle) when the first rotation axis is turned in tool tip center rotation handle feed or interrupt. (For settings, see "Table of correspondence with the manual handle feed axis selection signals" in the description of parameter No. 12310.)
The handle for which the signal states are set is determined by parameter No. 12323.

12314	States of the manual handle feed axis selection signals when the second rotation axis is turned in tool tip center rotation handle feed/interrupt

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 24

This parameter sets the states of the manual handle feed axis selection signals (HS1A to HS1E <Gn018.0 to Gn018.3, Gn411.0> for the first manual handle) or the manual handle interrupt axis selection signals (HS1IA to HS1IE <Gn041.0 to Gn041.3, Gn411.4> for the first manual handle) when the second rotation axis is turned in tool tip center rotation handle feed or interrupt. (For settings, see "Table of correspondence with the manual handle feed axis selection signals" in the description of parameter No. 12310.)
The handle for which the signal states are set is determined by parameter No. 12323.

12318	Tool length in 3-dimensional machining manual feed

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets a tool length when tool tip center rotation feed is performed with the 3-dimensional machining manual feed function and when the 3-dimensional machining manual feed screen is displayed.

NOTE
Specify a radius value to set this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
12319								CAC

[Input type] Parameter input
[Data type] Bit path

#0 CAC If a workpiece coordinate system offset is set for the rotation axis, the coordinate system of the rotation axis used to calculate the 3-dimensional manual feed is:

0: Machine coordinate system.
For those parameters Nos. 19680 to 19714 used to configure the machine that depend on the coordinates of the rotation axis, set the values assumed when the machine coordinates of the rotation axis are 0.

1: Workpiece coordinate system.
For those parameters Nos. 19680 to 19714 used to configure the machine that depend on the coordinates of the rotation axis, set the values assumed when the workpiece coordinates of the rotation axis are 0.

	#7	#6	#5	#4	#3	#2	#1	#0
12320	EM4					JFR	FLL	TWD

[Input type] Setting input
[Data type] Bit path

#0 TWD The directions of 3-dimensional machining manual feed (other than tool tip center rotation feed) when the tilted working plane indexing is issued are:

0: Same as those not in the tilted working plane indexing. That is, the directions are:
Tool axis normal direction 1 (table-based horizontal direction 1)
Tool axis normal direction 2 (table-based horizontal direction 2)
Tool axis direction (table-based vertical direction)

1: X, Y, and Z directions in the feature coordinate system.

#1 FLL The directions of tool axis normal direction feed or table-based horizontal direction feed in the 3-dimensional machining manual feed mode are:

0: Tool axis normal direction 1 (table-based horizontal direction 1) and tool axis normal direction 2 (table-based horizontal direction 2).

1: Longitude direction and latitude direction.

Bit 1 (FLL) of parameter No. 12320	Bit 0 (TWD) of parameter No. 12320	Directions of 3-dimensional machining manual feed
0	0	Conventional directions
0	1	When the tilted working plane indexing is issued: X, Y, and Z directions in the feature coordinate system When the command is not issued: Conventional directions
1	0	Longitude direction and latitude direction
1	1	When the tilted working plane indexing is issued: X, Y, and Z directions in the feature coordinate system When the command is not issued: Longitude direction and latitude direction

#2 JFR As the feeedrate of 3-dimensional machining manual feed (jog feed or incremental feed) :

0: The dry run rate (parameter No. 1410) is used.

1: The jog feedrate (parameter No. 1423) is used.

#7 EM4 Manual handle feed amount selection signal (incremental feed signal) MP4 <Gn019.6> is:

0: Disabled.

1: Enabled.

12321 **Normal axis direction**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3

When a tilted working plane indexing (G68.3) is issued to perform 3-dimensional machining manual feed in the latitude direction, longitude direction, and tool axis direction, this parameter sets an axis parallel to the normal direction.

1 : Positive (+) X-axis direction
2 : Positive (+) Y-axis direction
3 : Positive (+) Z-axis direction
0 : Reference tool axis direction (parameter No. 19697)

12322 **Angle used to determine whether to assume the tool axis direction to be parallel to the normal direction (parameter No. 12321)**

[Input type] Parameter input
[Data type] Real path
[Unit of data] deg

[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 to 90
When a tilted working plane indexing (G68.3) is issued to perform 3-dimensional machining manual feed in the latitude direction, longitude direction, and tool axis direction, if the angle between the tool axis direction and normal direction (parameter No. 12321) is too small, the tool axis direction is assumed to be parallel to the normal direction (parameter No. 12321). This parameter sets the maximum angle at which the tool axis direction is assumed to be parallel to the normal direction.
When this parameter is set to 0 or a value outside the valid range, it is set to 1 degree.

12323	Number of a manual handle used for 3-dimensional machining manual feed

[Input type] Setting input
[Data type] Byte path
[Valid data range] 0 to 5
When 3-dimensional machining manual feed (handle feed) is performed, set the number of the manual handle to be used.
When the second or third manual handle is used for 3-dimensional machining manual feed, the option for manual handle feed with 2/3 handles is required.
When the fourth or fifth manual handle is used for 3-dimensional machining manual feed, the option for manual handle feed with 4/5 handles is required.
If 0 or the number of an unavailable handle is set, the first handle is assumed.

12600	Identification Number for synchronous, composite and superimposed control with program command

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0,1 to 32767
Set identification numbers that can be specified with P,Q addresses.
The axis whose identification number is 0 cannot become under synchronous /composite /superimposed control by CNC program.
The same identification number cannot be set to two or more axes through all paths.
When the same identification number is set, alarm PS5339 occurs at G50.4/G50.5/G50.6/G51.4/G51.5/G51.6 block.

12605	Minimum waiting synchronous start M code in superimposed control for high-speed cycle machining

[Input type] Parameter input
[Data type] 2-word
[Valid data range] 0, 100 to 99999999
This parameter sets the waiting synchronous start M code to use if high-speed cycle machining or axis moving due to high-speed binary operation is to be started in synchronization with the superimposition command for an arbitrary operation path. For the waiting synchronous start M code, specify an M code that causes a synchronous start in the range of waiting M codes (parameters Nos. 8110 to 8111).
Set in this parameter the minimum M code that causes a synchronous start.

NOTE
Maximum M code that causes a synchronous start is set to parameter No. 8111.

	#7	#6	#5	#4	#3	#2	#1	#0
13113					CFD			CLR

[Input type] Parameter input
[Data type] Bit path

#0 CLR Upon reset, the display of a travel distance by 3-dimensional machining manual feed is:
0: Not cleared.
1: Cleared.

#3 CFD As feedrate F, the 3-dimensional machining manual feed screen displays:
0: Composite feedrate at the linear axis/rotation axis control point.
1: Feedrate at the tool tip.

	#7	#6	#5	#4	#3	#2	#1	#0
13117	SQP	SQB		INT				

[Input type] Parameter input
[Data type] Bit path

#4 INT During a program restart, the interference check on cutter/tool nose radius compensation is:
0: Enabled.
1: Disabled.

#6 SQB A program restart with a block number specification is:
0: Enabled.
1: Disabled.

#7 SQP A program restart with the P type is:
0: Enabled.
1: Disabled.

13131	Group number for simultaneous display of multiple paths

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 10
This parameter sets a group for simultaneous display on one screen in a multi-path system.
The paths set to belong to the same group are displayed on one screen.
If the values for all paths are set to 0, the simultaneous multi-path display function is disabled.

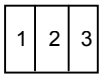
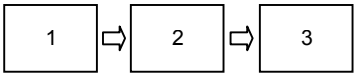
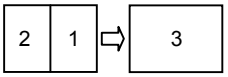
NOTE
When specifying groups, specify group numbers not less than 1 successively.
On 8.4-inch and 10.4-inch display units, up to three paths can be specified for simultaneous display.
On a 15-inch and 19-inch display units, up to four paths can be specified for simultaneous display.

13132	Simultaneous multi-path display order number

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to number of paths included in a simultaneous multi-path display group

This parameter sets the display order of a path defined to belong to a simultaneous multi-path display group.
Set the order, using numbers ranging from 1 to the number of paths included in the simultaneous multi-path display group.

[Example] Setting of simultaneous display group numbers and simultaneous display order numbers

Number of paths of CNC	Path	Display group number	Intra-group display order number	Screen display (Numbers represent displayed path numbers.)
One path	Path 1	1	1	
Three path	Path 1	1	1	
	Path 2	1	2	
	Path 3	1	3	
	Path 1	1	1	
	Path 2	2	1	
	Path 3	3	1	
	Path 1	1	2	
	Path 2	1	1	
	Path 3	2	1	

NOTE
Specify successive order numbers not less than 1 for the paths defined to belong to a group.

13151	SERIAL NUMBER

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 9999

SERIAL NUMBER is the number for the files output to a memory card by external output command DPRNT/BPRNT. 1 is added to SERIAL NUMBER when POPEN is executed. SERIAL NUMBER becomes 0 if it exceeds 9999.

NOTE
Please do not change the value of parameter No. 13151 because it is automatically updated.

	#7	#6	#5	#4	#3	#2	#1	#0
13200					ETE	TRT		

[Input type] Parameter input
[Data type] Bit path

#2 TRT As the remaining lifetime value for outputting the tool life arrival notice signal:
0: The remaining lifetime of the last tool is used.
1: The sum of the remaining lifetimes of the tools with the same type number is used.

NOTE
This parameter is valid when bit 3 (ETE) of parameter No. 13200 is set to 0 (arrival notice for each type number).

#3 ETE The tool life arrival notice signal is output:
0: For each tool type.
1: For each tool.

	#7	#6	#5	#4	#3	#2	#1	#0
13201							TDN	TDC

[Input type] Parameter input
[Data type] Bit

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

#0 TDC The function of customizing the tool management data screen of the tool management function is:
0: Disabled.
1: Enabled.

#1 TDN On the tool management function screen, the character string for indicating the tool life status can contain:
0: Up to 6 characters.
1: Up to 12 characters.

	#7	#6	#5	#4	#3	#2	#1	#0
13202	DOM	DOT		DO2	DOB	DOY	DCR	

[Input type] Parameter input
[Data type] Bit

#1 DCR On the tool management function screen, tool nose radius compensation data is:
0: Displayed.
1: Not displayed.

NOTE
This parameter is valid when the machine control type is the lathe system or compound system.

#2 DOY On the tool management function screen, Y-axis offset data is:
0: Displayed.
1: Not displayed.

NOTE
This parameter is valid when the machine control type is the lathe system or compound system.

#3 DOB On the tool management function screen, B-axis offset data is:
0: Displayed.
1: Not displayed.

> **NOTE**
> This parameter is valid when the machine control type is the lathe system or compound system.

#4 DO2 On the tool management function screen, the second geometry tool offset data is:
0: Displayed.
1: Not displayed.

> **NOTE**
> This parameter is valid when the machine control type is the lathe system or compound system.

#6 DOT On the tool management function screen, the tool offset data (X, Z) of the T series is:
0: Displayed.
1: Not displayed.

> **NOTE**
> This parameter is valid when the machine control type is the lathe system or compound system.

#7 DOM On the tool management function screen, the tool offset data of the M series is:
0: Displayed.
1: Not displayed.

> **NOTE**
> This parameter is valid when the machine control type is the machining center system or compound system.

	#7	#6	#5	#4	#3	#2	#1	#0
13204								TDL

[Input type] Parameter input
[Data type] Bit

#0 TDL The protection function for tool management data using a key is:
0: Disabled.
1: Enabled.

13220	Number of valid tools in tool management data

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 64 (Extended to 240 or 1000 by the addition of an option)
This parameter sets the number of valid tools in tool management data.

13221	M code for tool life count restart of tool management or tool life management

[Input type] Parameter input
[Data type] Word path
[Valid data range]

- When tool management function is used:
  When 0 is set in this parameter, this parameter is ignored.
  When an M code for tool life count restart is specified, the counting of the life of the tool attached at the spindle position is started. When the type for counting the number of use times is selected, the target of life counting is switched to the tool attached at the spindle position, and the life count is incremented by 1.
  When the type for counting time is selected, the target of life counting is switched to the tool attached at the spindle position but no other operations are performed. If the tool attached at the spindle position is not a tool under tool life management, no operation is performed.
  The M code set in parameter No. 6811 waits for FIN. However, the M code set in this parameter does not wait for FIN.
  The M code set in parameter No. 13221 must not be specified in a block where another auxiliary function is specified.
  The M code set in parameter No. 13221 does not wait for FIN. So, do not use the M code for other purposes.
  Valid data range is 0 to 65535.
- When tool life management function is used:
  When 0 is set, this parameter is ignored.
  For the operation of an M code for tool life count restart, see the description of parameter No. 6811.
  This parameter is used when an M code for tool life count restart exceeds 127.
  Set parameter No. 6811 to 0, and set the value of an M code in this parameter.
  Valid data range is 0 to 255 (not including 01, 02, 30, 98, and 99).

13222	Number of data items in the first cartridge

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 64 (Extended to 240 or 1000 by the addition of an option)
This parameter sets the number of data items used with the first cartridge.

13223	Start pot number of the first cartridge

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 9999
This parameter sets the start pot number to be used with the first cartridge. Pot numbers starting with the value set in this parameter and sequentially incremented by 1 are assigned to all data items.

13227	Number of data items in the second cartridge

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 64(Extended to 240 or 1000 by the addition of an option)
This parameter sets the number of data items used with the second cartridge.

13228	Start pot number of the second cartridge

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 9999
This parameter sets the start pot number to be used with the second cartridge. Pot numbers starting with the value set in this parameter and sequentially incremented by 1 are assigned to all data items.

13232	Number of data items in the third cartridge

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 64(Extended to 240 or 1000 by the addition of an option)
This parameter sets the number of data items used with the third cartridge.

13233	Start pot number of the third cartridge

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 9999
This parameter sets the start pot number to be used with the third cartridge. Pot numbers starting with the value set in this parameter and sequentially incremented by 1 are assigned to all data items.

13237	Number of data items in the fourth cartridge

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 64(Extended to 240 or 1000 by the addition of an option)
This parameter sets the number of data items used with the fourth cartridge.

13238	Start pot number of the fourth cartridge

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 9999
This parameter sets the start pot number to be used with the fourth cartridge. Pot numbers starting with the value set in this parameter and sequentially incremented by 1 are assigned to all data items.

13252	M code for specifying a particular tool

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 65535
This parameter sets not a tool type number but an M code for directly specifying the T code of a particular tool.

13265	Number for selecting a spindle position offset number in tool management or H code for using the tool length offset in tool life management

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 9999

- When tool management function is used:
  This parameters sets an H/D code for selecting an offset number registered in the data of the tool attached at the spindle position.
  When 0 is set, an ordinary used code such as H99/D99 is used. When a value other than 0 is set, H99/D99 no longer has a particular meaning. So, when H99/D99 is specified in this case, the specification of offset number 99 is assumed.
  With the T series, address D only is used to specify a tool number and offset number, so that a restriction is imposed on the number of digits. So, the valid data range of this parameter varies according the number of digits of an offset number.
  When the number of digits of an offset number is 1: to 9
  When the number of digits of an offset number is 2: to 99
  When the number of digits of an offset number is 3: to 999

- When tool life management function is used:
  Usually, when H99 is specified, tool length offset is enabled by the H code of the tool being used. By setting any H code in this parameter, the H code instead of H99 can be used. If 0 is specified, H99 is assumed.
  A value ranging from 0 to 9999 can be set.

13266	D code for enabling cutter compensation in tool life management

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 9999
Usually in tool life management, specifying D99 allows the D code of the tool being used to enable cutter compensation. By setting any D code in this parameter, the D code instead of D99 can be used. If 0 is set, D99 is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
13421						FRF	FCN	FRS

[Input type] Parameter input
[Data type] Bit path

#0 FRS In a program containing an M code for turning the flexible synchronous mode on/off, a program restart is:
0: Disabled.
1: Enabled.

#1 FCN In the emergency stop/servo off state, Inter-Path flexible synchronous control is:
0: Canceled.
1: Not canceled.

#2 FRF If G27/G28/G29/G30/G30.1/G53 is specified during flexible synchronous control, alarm PS0010, “IMPROPER G-CODE” is:
0: Issued.
1: Is not issued. Commands to the master axis are possible.
Even if, however, parameter bit FRF is set to 1, and G28 is specified for the master axis in the state in which the reference position of the master axis subject to flexible synchronous control is not established, or if G27/G28/G29/G30/G30.1/G53 is specified for the slave axis, alarm PS5381, “INVALID COMMAND IN FSC MODE” is issued.

NOTE
If the option for inter-path flexible synchronous control is specified, even if 0 is set in the parameter bit FRF, the operation will be the same as that if 1 is set.

	#7	#6	#5	#4	#3	#2	#1	#0
13450				MFC				

[Input type] Parameter input
[Data type] Bit path

#4 MFC When the cutting is executed without specifying a feedrate (F) after the modal G code of group 05 was changed by G93(inverse time feed) / G94(feed per minute) / G95(feed per revolution) command,

0: The feedrate (F) is inherited as a modal.
1: Alarm PS0011, “FEED ZERO (COMMAND)” is issued.

NOTE
1 In G93 mode, if the axis command and the feedrate (F) command are not in the same block, alarm PS1202, "NO F COMMAND AT G93" is issued regardless of the setting of this parameter.
2 If this parameter bit is set to 1, and if the G code of group 05 is cleared due to a reset, by setting bit 6 (CLR) of parameter No. 3402 to 1 and bit 5 (C05) of parameter No. 3406 to 0, so that the modal G code is switched, the feedrate (F) will be cleared even if bit 7 (CFH) of parameter No. 3409 is set to 1.
3 If this parameter bit is 1, and bit 7 (FC0) of parameter No. 1404 is set to 1, alarm PS0011 is not issued and the block is executed with a feedrate of 0 even if the feed selection command is used to switch the modal code of group 05 and the axis command is executed in cutting feed mode without specifying a feedrate (F). In G93 mode, alarm PS1202 is issued regardless of the setting of the parameter bit FC0.
4 If this parameter bit is 1, alarm PS0011 or PS1202 is not used even if the feed selection command is used to switch the modal code of group 05 and the axis command is executed in cutting feed mode without specifying a feedrate (F), provided that the travel distance is 0.
5 If this parameter bit is 1, alarms PS0011 and PS1202 are issued if the feed selection command is used to switch the modal code of group 05 and the axis command is executed in cutting feed mode without specifying a feedrate (F), even if cutting feedrate (parameter No. 1411) during automatic operation is set. (This is true of the M series.)

	#7	#6	#5	#4	#3	#2	#1	#0
13451							ATW	

[Input type] Parameter input
[Data type] Bit path

#1 ATW When I, J, and K are all set to 0 in a block that specifies a feature coordinate system setup command (G68.2), which is a tilted working plane command:

0: An alarm PS5457, “G68.2 FORMAT ERROR” is issued.
1: A feature coordinate system with a tilted plane angle of 0 degrees is assumed for operation.

	#7	#6	#5	#4	#3	#2	#1	#0
13600	MSA							MCR

[Input type] Parameter input
[Data type] Bit path

#0 MCR When an allowable acceleration rate adjustment is made with the machining condition selection function or machining quality level adjustment function (machining parameter adjustment screen, precision level selection screen), parameter No. 1735 for the deceleration function based on acceleration in circular interpolation is:
0: Modified.
1: Not modified.

#7 MSA When the machining condition selection function or machining quality level adjustment function is used, the acceleration rate change time (bell-shaped) (LV1, LV10) is:
0: Set using parameter Nos. 13612 and 13613.
1: Set using parameter Nos. 13662 and 13663.

	#7	#6	#5	#4	#3	#2	#1	#0
13601								MPR

[Input type] Parameter input
[Data type] Bit

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 MPR The machining parameter adjustment screen is:
0: Displayed.
1: Not displayed.
Even if 1 is set in this parameter bit, the precision level selection screen for the machining condition selecting function and the precision level selection screens (machining quality level selection screen and the machining level selection screen) for the machining quality level adjustment function are displayed.

13610	Acceleration rate for acceleration/deceleration before look-ahead interpolation in AI contour control (precision level 1)
13611	Acceleration rate for acceleration/deceleration before look-ahead interpolation in AI contour control (precision level 10)

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0)
Each of these parameters sets an acceleration rate for acceleration/ deceleration before interpolation in AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13612	Acceleration rate change time (bell-shaped) when AI contour control is used (precision level 1)
13613	Acceleration rate change time (bell-shaped) when AI contour control is used (precision level 10)

[Input type] Parameter input
[Data type] Byte path
[Unit of data] msec
[Valid data range] 0 to 127

Each of these parameters sets an acceleration rate change time (bell-shaped) in AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13614	**Allowable acceleration rate change amount for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration (precision level 1)**
13615	**Allowable acceleration rate change amount for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration (precision level 10)**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0)
Each of these parameters sets an allowable acceleration rate change amount per 1 ms for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration during AI contour control.
Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13616	**Allowable acceleration rate change amount for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration in successive linear interpolation operations (precision level 1)**
13617	**Allowable acceleration rate change amount for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration in successive linear interpolation operations (precision level 10)**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0)
Each of these parameters sets an allowable acceleration rate change amount per 1 ms for each axis in speed control based on acceleration rate change under control on the rate of change of acceleration in successive linear interpolation operations during AI contour control.
Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

NOTE
1 For an axis with 0 set in this parameter, parameter No. 13614 and No. 13615 (allowable acceleration rate change amount in speed control based on acceleration rate change under control on the rate of change of acceleration) are valid.
2 For an axis with 0 set in parameter No. 13614 and No. 13615 (allowable acceleration rate change amount in speed control based on acceleration rate change under control on the rate of change of acceleration), speed control based on acceleration rate change is disabled, so that the specification of this parameter has no effect.

13618	Rate of change time of the rate of change of acceleration in smooth bell-shaped acceleration/deceleration before interpolation when AI contour control is used (precision level 1)
13619	Rate of change time of the rate of change of acceleration in smooth bell-shaped acceleration/deceleration before interpolation when AI contour control is used (precision level 10)

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 50

Each of these parameters sets the rate (percentage) of the change time of the rate of change of acceleration to the change time of acceleration rate change in smooth bell-shaped acceleration/deceleration before look-ahead interpolation during AI contour control.
Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

NOTE
When 0 or a value not within the valid data range is set in this parameter, smooth bell-shaped acceleration/deceleration before look-ahead interpolation is not performed.

13620	Allowable acceleration rate when AI contour control is used (precision level 1)
13621	Allowable acceleration rate when AI contour control is used (precision level 10)

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0)
Each of these parameters sets an allowable acceleration rate in AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13622	Time constant for acceleration/deceleration after interpolation when AI contour control is used (precision level 1)
13623	Time constant for acceleration/deceleration after interpolation when AI contour control is used (precision level 10)

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 1 to 512

Each of these parameters sets a time constant for acceleration/ deceleration after interpolation when AI contour control is used. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13624	Corner speed difference when AI contour control is used (precision level 1)
13625	Corner speed difference when AI contour control is used (precision level 10)

[Input type] Parameter input

[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Each of these parameters sets an allowable speed difference for speed determination based on corner speed difference in AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13626	**Maximum cutting speed when AI contour control is used (precision level 1)**
13627	**Maximum cutting speed when AI contour control is used (precision level 10)**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Each of these parameters sets a maximum cutting speed in AI contour control. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

13628	**Parameter number corresponding to arbitrary item 1 when AI contour control is used**
13629	**Parameter number corresponding to arbitrary item 2 when AI contour control is used**

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 65535
These parameters set the parameter numbers corresponding to arbitrary items 1 and 2.

NOTE
The parameter numbers corresponding to the following cannot be specified:
- Bit parameters
- Spindle parameters Nos. 4000 to 4799
- Parameters of real number type
- Parameters that require power-off (for which the alarm PW0000, “POWER MUST BE OFF” is issued)
- Nonexistent parameters

13630	Value with emphasis on speed (precision level 1) of the parameter corresponding to arbitrary item 1 when AI contour control is used
13631	Value with emphasis on speed (precision level 1) of the parameter corresponding to arbitrary item 2 when AI contour control is used
13632	Value with emphasis on speed (precision level 10) of the parameter corresponding to arbitrary item 1 when AI contour control is used
13633	Value with emphasis on speed (precision level 10) of the parameter corresponding to arbitrary item 2 when AI contour control is used

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Depend on the type of parameter for an arbitrary item
[Valid data range] Depend on the type of parameter for an arbitrary item
Each of these parameters sets a value with emphasis placed on speed or precision for a parameter.

13662	Acceleration rate change time (bell-shaped) when AI contour control is used (precision level 1), range extended
13663	Acceleration rate change time (bell-shaped) when AI contour control is used (precision level 10), range extended

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] msec
[Valid data range] 0 to 200
Each of these parameters sets an acceleration rate change time (bell-shaped) in AI contour control. Set a value (precision 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

	#7	#6	#5	#4	#3	#2	#1	#0
14000						IRFx		

[Input type] Parameter input
[Data type] Bit axis

#2 IRFx An inch-metric switch command (G20, G21) at the reference position is:
0: Disabled.
1: Enabled.

When this function is enabled for an axis, if an attempt to switch between the inch and metric unit is made although the tool is not at the reference position on that axis, an alarm PS5362, "CONVERT INCH/MM AT REF-POS" is issued, and switching between the inch and metric unit is canceled.
Be sure to move the tool to the reference position by, for example, specifying G28 before switching between the inch and metric unit.

14010	Maximum allowable travel distance when the reference position is established for a linear scale with an absolute address reference position

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
This parameter sets the maximum allowable travel distance at the FL rate when the reference position is established for a linear scale with an absolute address reference position. When the travel distance exceeds the setting of this parameter, the alarm DS0017, "SERIAL DCL:REF-POS ESTABLISH ERR" is issued. When this parameter is set to 0, the maximum allowable travel distance is not checked.

NOTE
1 To establish the reference position with axis synchronous control, set the parameter for both master and slave axes.
2 In angular axis control, the setting of this parameter is invalid to the orthogonal axis where the reference position on the angular axis is being established.

	#7	#6	#5	#4	#3	#2	#1	#0
14250					RGE			

[Input type] Parameter input
[Data type] Bit path

#3 RGE The division of the gain multiplier of the pivot axis is:
0: Performed at up to 10 points.
For the pivot axis, set parameters Nos. 14270 to 14279 (angle) and parameters Nos. 14280 to 14289 (gain multiplier for the angle).
1: Expanded at up to (10 × number of controlled axes).
The method of changing parameters Nos. 14270 to 14279 (angle) and parameters Nos. 14280 to 14289 (gain multiplier for the angle) is changed, and the number of division points varies with the number of controlled axes.

NOTE
When this parameter is set, the power must be turned off before operation is continued.

When the bit 3 (RGE) of parameter No. 14250 is set to 0

14270	Angle 1 (θ - data for G diagrams)
14271	Angle 2 (θ - data for G diagrams)
14272	Angle 3 (θ - data for G diagrams)
14273	Angle 4 (θ - data for G diagrams)
14274	Angle 5 (θ - data for G diagrams)
14275	Angle 6 (θ - data for G diagrams)
14276	Angle 7 (θ - data for G diagrams)
14277	Angle 8 (θ - data for G diagrams)
14278	Angle 9 (θ - data for G diagrams)
14279	Angle 10 (θ - data for G diagrams)

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
When the increment system is IS-B, 0.0 to +360.
Set these parameters for the pivot axis.

14280	Setting for the gain multiplier of angle 1 (θ - gain for G diagrams)
14281	Setting for the gain multiplier of angle 2 (θ - gain for G diagrams)
14282	Setting for the gain multiplier of angle 3 (θ - gain for G diagrams)
14283	Setting for the gain multiplier of angle 4 (θ - gain for G diagrams)
14284	Setting for the gain multiplier of angle 5 (θ - gain for G diagrams)
14285	Setting for the gain multiplier of angle 6 (θ - gain for G diagrams)
14286	Setting for the gain multiplier of angle 7 (θ - gain for G diagrams)
14287	Setting for the gain multiplier of angle 8 (θ - gain for G diagrams)
14288	Setting for the gain multiplier of angle 9 (θ - gain for G diagrams)
14289	Setting for the gain multiplier of angle 10 (θ - gain for G diagrams)

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] ×1/512
[Min. unit of data] -32768(×-63) to 32767(×64.9)
Set these parameters for the pivot axis.
The gain multiplier is calculated using the setting of a parameter, as follows:

$$\text{Gain multiplier} = \frac{\text{Setting of parameter}}{512} + 1$$

Thus, the gain multipliers for the settings below are as given below.

Setting	-32768	-1536	-1024	512	1024	1536	32767
Gain multiplier	-63	-2	-1	2	3	4	64.9

When the bit 3 (RGE) of parameter No. 14250 is set to 1

14270	Angle (θ - gain for G diagrams)
to	to
14279	Angle (θ - gain for G diagrams)

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Real axis
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] -1, 0, or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, -1.0, 0.0 to +360.)
[Example] Set angles as follows:
1st axis: Angles 1.0 to 10.0
2nd axis: Angles 11.0 to 20.0
3rd axis: Angles 21.0 to 30.0
to
8th axis: Angles 71.0 to 75.0

Parameter	1st axis	2nd axis	3rd axis	4th axis	5th axis	6th axis	7th axis	8th axis
14270	1.0	11.0	21.0	31.0	41.0	51.0	61.0	71.0
14271	2.0	12.0	22.0	32.0	42.0	52.0	62.0	72.0
14272	3.0	13.0	23.0	33.0	43.0	53.0	63.0	73.0
14273	4.0	14.0	24.0	34.0	44.0	54.0	64.0	74.0
14274	5.0	15.0	25.0	35.0	45.0	55.0	65.0	75.0
14275	6.0	16.0	26.0	36.0	46.0	56.0	66.0	-1.0
14276	7.0	17.0	27.0	37.0	47.0	57.0	67.0	—
14277	8.0	18.0	28.0	38.0	48.0	58.0	68.0	—
14278	9.0	19.0	29.0	39.0	49.0	59.0	69.0	—
14279	10.0	20.0	30.0	40.0	50.0	60.0	70.0	—

Set -1.0 in the parameter for "maximum number of items used + 1", where items refer to angles.
The table gives values if angles 1.0 to 75.0 are set in 1-degree steps.
If there are multiple pivot axes, the settings are used universally to all the pivot axes.

14280	**Setting for the gain multiplier of angle (θ - gain for G diagrams)**
to	**to**
14289	**Setting for the gain multiplier of angle (θ - gain for G diagrams)**

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] ×1/512
[Min. unit of data] -32768(×-63) to 32767(×64.9)

The gain multiplier is calculated using the setting of a parameter, as follows:

$$\text{Gain multiplier} = \frac{\text{Setting of parameter}}{512} + 1$$

Thus, the gain multipliers for the settings below are as given below.

Setting	-32768	-1536	-1024	512	1024	1536	32767
Gain multiplier	-63	-2	-1	2	3	4	64.9

[Example] Set the settings for the gain multipliers for angles (parameters Nos. 14270 to 14279) as follows:
1st axis: Setting for the gain multiplier for angles 1.0 to 10.0
2nd axis: Setting for the gain multiplier for angles 11.0 to 20.0
3rd axis: Setting for the gain multiplier for angles 21.0 to 30.0
to
8th axis: Setting for the gain multiplier for angles 71.0 to 75.0

Parameter	1st axis	2nd axis	3rd axis	4th axis	5th axis	6th axis	7th axis	8th axis
14280	614	1382	1843	2150	2150	2048	1945	1782
14281	691	1428	1894	2151	2140	2038	1928	1763
14282	768	1474	1920	2152	2130	2027	1912	1743
14283	845	1520	1945	2153	2119	2017	1896	1724
14284	922	1566	1971	2154	2109	2007	1880	1704
14285	999	1612	1997	2155	2099	1996	1864	−
14286	1076	1659	2022	2154	2089	1986	1847	−
14287	1153	1705	2048	2153	2078	1976	1831	−
14288	1230	1751	2073	2152	2068	1966	1815	−
14289	1307	1797	2099	2151	2058	1955	1798	−

In this example, the 12th angle is set for the second axis of parameter No. 14271, and the setting for the gain multiplier for the angle is set for the second axis of parameter No. 14281.

NOTE
1 When bit 3 (RGE) of parameter No. 14250 is set to 1, the number of angles and the number of settings for the gain multipliers for the angles vary depending on the number of controlled axes.
[Example]
For eight axes, up to 80 items can be set, and for four axes, up to 40 items can be set.
2 If there are multiple pivot axes, the settings are used universally to all the pivot axes.

14713	Unit of magnification by which enlargement and reduction is performed with the dynamic graphic display function

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 255
This parameter sets the unit of magnification by which enlargement and reduction is performed with the dynamic graphic display function.
Unit of magnification = 64 / setting
If 0 is set, 64 is assumed.

14714	Unit of horizontal movement when a movement is made with the dynamic graphic display function

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 255
This parameter sets the unit of horizontal movement (in dots) applied when a movement is made with the dynamic graphic display function.
If 0 is set, 64 is assumed.

14715	Unit of vertical movement when a movement is made with the dynamic graphic display function

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 255
This parameter sets the unit of vertical movement (in dots) applied when a movement is made with the dynamic graphic display function.
If 0 is set, 35 is assumed.

14716	Unit of rotation angle when rotation is performed with the dynamic graphic display function

[Input type] Parameter input
[Data type] Word
[Valid data range] 0 to 255
This parameter sets the unit (in degrees) of a rotation angle by which the drawing coordinate system is rotated with the dynamic graphic display function.
If 0 is set, 10 is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
18000							RTW	

[Input type] Parameter input
[Data type] Bit

#1 RTW At the start of a re-forward movement operation of the manual handle retrace function in a multi-path system,

0: The re-forward movement operation is performed immediately on each path.

1: Those paths for which reverse movement is prohibited are synchronized at the stop position.

18060	M code that prohibits backward movement

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 999

When an M code that prohibits backward movement is specified during backward movement, backward movement of blocks before the M code is prohibited. In this case, backward movement prohibition signal MRVSP<Fn091.2> is output.

The M code that prohibits backward movement is not output to the PMC as an M code. As the M code that prohibits backward movement, set an M code that is not used by auxiliary functions and macros.

18065	M code 1 that prohibits backward movement and is output as an M code

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 999

When an M code that prohibits backward movement is specified during backward movement, backward movement of blocks before the M code is prohibited. In this case, backward movement prohibition signal MRVSP<Fn091.2> is output.

18066	M code 2 that prohibits backward movement and is output as an M code

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 999

When an M code that prohibits backward movement is specified during backward movement, backward movement of blocks before the M code is prohibited. In this case, backward movement prohibition signal MRVSP<Fn091.2> is output.

	#7	#6	#5	#4	#3	#2	#1	#0
19500		FNW						

[Input type] Parameter input
[Data type] Bit path

#6 FNW When the feedrate is determined according to the feedrate difference and acceleration in AI contour control:

0: The maximum feedrate at which the allowable feedrate difference and acceleration for each axis are not exceeded is used.

1: A feedrate is determined to satisfy the condition that the allowable feedrate difference and allowable acceleration rate of each axis are not exceeded, and also to ensure that a constant deceleration rate is applied to the same figure regardless of the direction of movement.

	#7	#6	#5	#4	#3	#2	#1	#0
19501			FRP					

[Input type] Parameter input
[Data type] Bit path

#5 FRP Linear rapid traverse is:

0: Acceleration/deceleration after interpolation

1: Acceleration/deceleration before interpolation

Set a maximum allowable acceleration rate for each axis in parameter No. 1671.
When using bell-shaped acceleration/deceleration before interpolation, set an acceleration rate change time in parameter No. 1672.

When this parameter is set to 1, acceleration/deceleration before interpolation is also applied to rapid traverse if all conditions below are satisfied. At this time, acceleration/deceleration after interpolation is not applied.

- Bit 1 (LRP) of parameter No. 1401 is set to 1: Linear interpolation type positioning
- A value other than 0 is set in parameter No. 1671 for an axis.
- The AI contour control mode is set.

If all of these conditions are not satisfied, acceleration/deceleration after interpolation is applied.

	#7	#6	#5	#4	#3	#2	#1	#0
19503				ZOL				HPF

[Input type] Parameter input
[Data type] Bit path

#0 HPF When a feedrate is determined based on acceleration in AI contour control, smooth feedrate control is:

0: Not used.

1: Used.

#4 ZOL The deceleration function based on cutting load in AI contour control (deceleration based on Z-axis fall angle) is:

0: Enabled for all commands.

1: Enabled for linear interpolation commands only.

	#7	#6	#5	#4	#3	#2	#1	#0
19515							ZG2	

[Input type] Parameter input
[Data type] Bit path

#1 ZG2 When the deceleration function based on cutting load in AI contour control (deceleration based on Z-axis fall angle) is used:
0: Stepwise override values are applied.
1: Inclined override values are applied.
This parameter is valid only when bit 4 (ZAG) of parameter No. 8451 is set to 1.
When this parameter is set to 1, be sure to set parameter Nos. 19516, 8456, 8457, and 8458.

19516	Override for area 1 in deceleration based on cutting load in AI contour control

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 100
This parameter sets an override value for area 1 when the deceleration function based on cutting load in AI contour control is used.
This parameter is valid only when bit 1 (ZG2) of parameter No. 19515 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
19530		CYS						

[Input type] Parameter input
[Data type] Bit path

#6 CYS Specifies whether when the cylindrical interpolation cutting point compensation function is used, cutting point compensation is performed between blocks or together with a block movement if the cutting point compensation value is less than the setting of parameter No. 19534.
0: Performed between blocks.
1: Performed together with a block movement if the cutting point compensation value is less than the setting of parameter No. 19534.

19534	Limit for changing cylindrical interpolation cutting point compensation in a single block

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 1 to 999999999
The following operation is performed, depending on the setting of parameter No. 19530:
(1) Bit 6 (CYS) of parameter No. 19530) is set to 0
If the amount of cylindrical interpolation cutting point compensation is smaller than the value set in this parameter, cylindrical interpolation cutting point compensation is not performed. Instead, this ignored amount of cylindrical interpolation cutting point compensation is added to the next amount of cylindrical interpolation cutting point compensation to determine whether to perform cylindrical interpolation cutting point compensation.

(2) Bit 6 (CYS) of parameter No. 19530) is set to 1
If the amount of cylindrical interpolation cutting point compensation is smaller than the value set in this parameter, cylindrical interpolation cutting point compensation is performed together with the movement of the specified block.

NOTE
Set this parameter as follows:
Setting < (setting for a rotation axis in parameter No. 1430) × 4/3
where 4/3 is a constant for internal processing.

19535	Limit of travel distance moved with the cylindrical interpolation cutting point compensation in the previous block unchanged.

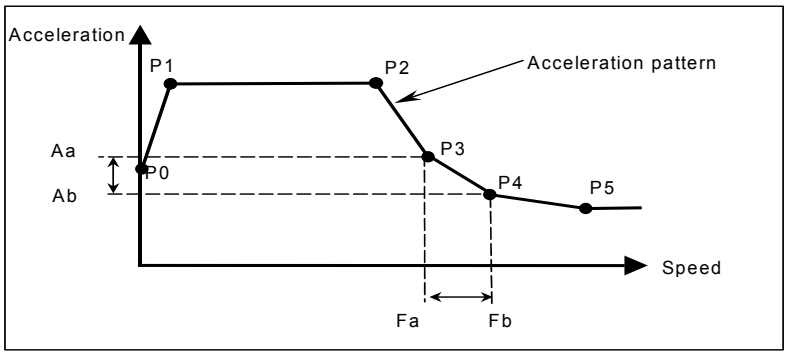
[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 1 to 999999999
The following operation is performed, depending on the type of interpolation:
(1) For linear interpolation
If the travel distance in a specified block is smaller than the value set in this parameter, machining is performed without changing the cylindrical interpolation cutting point compensation in the previous block.
(2) For circular interpolation
If the diameter of a specified arc is smaller than the value set in this parameter, machining is performed without changing the cylindrical interpolation cutting point compensation in the previous block. Cylindrical interpolation cutting point compensation is not performed according to a circular movement.

	#7	#6	#5	#4	#3	#2	#1	#0
19540								FAP

[Input type] Parameter input
[Data type] Bit path

#0 FAP Optimal torque acceleration/deceleration is:
0: Disabled.
1: Enabled.
When the linear positioning parameters, namely bit 1 (LRP) of parameter No. 1401 and bit 0 (FAP) of parameter No. 19540, are set to 1, and a value other than 0 is set in reference acceleration parameter No. 1671 for an axis, the acceleration/deceleration for rapid traverse becomes optimal torque acceleration/deceleration in the mode for acceleration/deceleration before look-ahead interpolation (or the AI contour control mode). Optimal torque acceleration/ deceleration is controlled according to parameter-set restricted acceleration curve data.

Setting an acceleration pattern



Set the speed at each of the acceleration setting points (P0 to P5) in a corresponding parameter, then in parameters for each axis, set acceleration rates applicable in the following four cases at these speeds: when a movement in the positive direction is accelerated, when a movement in the positive direction is decelerated, when a movement in the negative direction is accelerated, and when a movement in the negative direction is decelerated.
The line connecting the acceleration setting points is the acceleration pattern.
The acceleration rate for each axis is calculated. For example, between speeds Fa to Fb in the above figure, the acceleration rates corresponding to these speeds, Aa to Ab, are used for calculation.
The tangent acceleration is controlled so that it does not exceed the calculated acceleration rate for each axis.

> ⚠ **CAUTION**
> When an acceleration pattern is set, setting a high acceleration rate immediately after a speed of 0 can cause an impact on the machine, so it is not desirable. Therefore, **be sure to apply a relatively low acceleration rate at a speed of 0** as shown in the above figure.

19541	**Optimal torque acceleration/deceleration (speed at P1)**
19542	**Optimal torque acceleration/deceleration (speed at P2)**
19543	**Optimal torque acceleration/deceleration (speed at P3)**
19544	**Optimal torque acceleration/deceleration (speed at P4)**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 0.01%
[Valid data range] 0 to 10000

The speeds at acceleration setting points P1 to P4 are to be set with speed parameters Nos. 19541 to 19544 as ratios to the rapid traverse speed (parameter No. 1420). The speed at P0 is 0, and the speed at P5 is the rapid traverse rate specified with parameter No. 1420. Any acceleration setting point for which the speed parameter (one of Nos. 19541 to 19544) is set to 0 will be skipped.

19545	Optimal torque acceleration/deceleration (acceleration at P0 during movement in + direction and acceleration)
19546	Optimal torque acceleration/deceleration (acceleration at P1 during movement in + direction and acceleration)
19547	Optimal torque acceleration/deceleration (acceleration at P2 during movement in + direction and acceleration)
19548	Optimal torque acceleration/deceleration (acceleration at P3 during movement in + direction and acceleration)
19549	Optimal torque acceleration/deceleration (acceleration at P4 during movement in + direction and acceleration)
19550	Optimal torque acceleration/deceleration (acceleration at P5 during movement in + direction and acceleration)
19551	Optimal torque acceleration/deceleration (acceleration at P0 during movement in - direction and acceleration)
19552	Optimal torque acceleration/deceleration (acceleration at P1 during movement in - direction and acceleration)
19553	Optimal torque acceleration/deceleration (acceleration at P2 during movement in - direction and acceleration)
19554	Optimal torque acceleration/deceleration (acceleration at P3 during movement in - direction and acceleration)
19555	Optimal torque acceleration/deceleration (acceleration at P4 during movement in - direction and acceleration)
19556	Optimal torque acceleration/deceleration (acceleration at P5 during movement in - direction and acceleration)
19557	Optimal torque acceleration/deceleration (acceleration at P0 during movement in + direction and deceleration)
19558	Optimal torque acceleration/deceleration (acceleration at P1 during movement in + direction and deceleration)
19559	Optimal torque acceleration/deceleration (acceleration at P2 during movement in + direction and deceleration)
19560	Optimal torque acceleration/deceleration (acceleration at P3 during movement in + direction and deceleration)
19561	Optimal torque acceleration/deceleration (acceleration at P4 during movement in + direction and deceleration)
19562	Optimal torque acceleration/deceleration (acceleration at P5 during movement in + direction and deceleration)
19563	Optimal torque acceleration/deceleration (acceleration at P0 during movement in - direction and deceleration)
19564	Optimal torque acceleration/deceleration (acceleration at P1 during movement in - direction and deceleration)
19565	Optimal torque acceleration/deceleration (acceleration at P2 during movement in - direction and deceleration)
19566	Optimal torque acceleration/deceleration (acceleration at P3 during movement in - direction and deceleration)
19567	Optimal torque acceleration/deceleration (acceleration at P4 during movement in - direction and deceleration)

19568	Optimal torque acceleration/deceleration (acceleration at P5 during movement in - direction and deceleration)

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 0.01%
[Valid data range] 0 to 32767
For each travel direction and each acceleration/deceleration operation, set the allowable acceleration rate at each of the acceleration setting points (P0 to P5). As an allowable acceleration rate, set a ratio to the value set in the reference acceleration parameter No. 1671. When 0 is set, the specification of 100% is assumed.

19581	Tolerance smoothing for nano smoothing

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch, degree (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets a tolerance value for a program created using miniature line segments in nano smoothing.
When 0 is set in this parameter, a minimum amount of travel in the increment system is regarded as a tolerance value.

19582	Minimum amount of travel of a block that makes a decision based on an angular difference between blocks for nano smoothing

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch, degree (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the minimum amount of travel of a block that makes a decision based on an angular difference between blocks for nano smoothing. A block that specifies an amount of travel less than the value set in this parameter makes no decision based on an angular difference.
When 0 is set in this parameter, a decision based on an angular difference is made with all blocks.
A value greater than the value set in parameter No. 8490 for making a decision based on the minimum travel distance of a block must be set.

19587	Tolerance of rotary axes for nano smoothing 2

[Input type] Setting input
[Data type] Real axis
[Unit of data] degree (input unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 - +999999.999)

This parameter sets the tolerance of rotation axes in a program created using small line segments in nano smoothing 2. This parameter is valid only for the rotation axes specified in nano smoothing 2.
When 0 is set in this parameter, a minimum amount of travel in the increment system is regarded as a tolerance value.

	#7	#6	#5	#4	#3	#2	#1	#0
19604								TPC

[Input type] Setting input
[Data type] Bit path

#0 TPC In the case that there is no address P at the start of tool center point control (G43.4/G43.5), tool posture control
0: Does not work.
1: Works.

	#7	#6	#5	#4	#3	#2	#1	#0
19605	TIT							NSC

[Input type] Parameter input
[Data type] Bit path

#0 NSC For the machine type that has no rotation axis for rotating the tool (when parameter No. 19680 is set to 12 to specify the table rotation type), control point shifting in the tilted working plane command is:
0: Enabled.
Set bit 4 (SPR) and bit 5 (SVC) of parameter No. 19665.
1: Disabled.

#7 TIT If, in a tool rotary type machine (parameter No. 19680 = 2), tool center point control and inverse time feed or feed per revolution are used together,
0: Inverse time feed or feed per revolution is applied during tool center point control.
1: It operates as tool length compensation in tool axis direction.

NOTE
For the composite and table rotary types, inverse time feed or feed per revolution is applied during tool center point control regardless of this parameter.

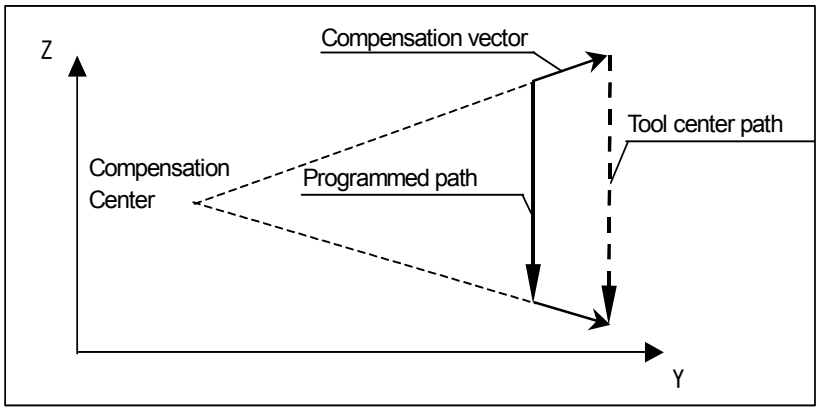
	#7	#6	#5	#4	#3	#2	#1	#0
19607			CAV				SPG	
			CAV		WCD		SPG	

[Input type] Parameter input
[Data type] Bit path

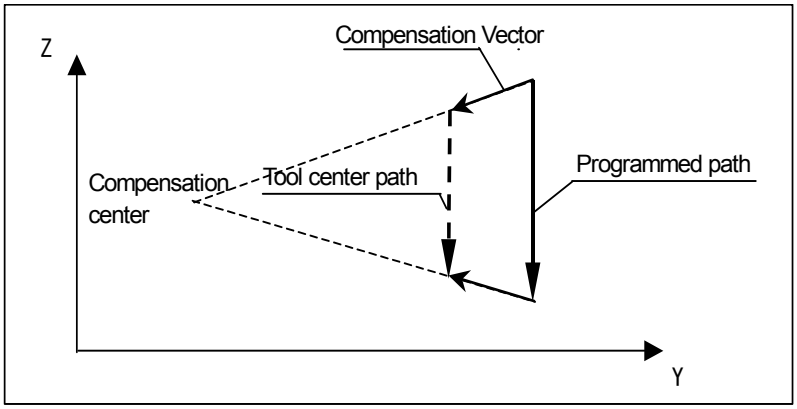
#1 SPG To apply 3-dimensional tool compensation to a machine having a table rotation axis, as the G code to be specified:
0: G41.2/G42.2 is used regardless of the machine type.
1: G41.4/G42.4 is used for a table rotation type machine; G41.5/G42.5 for a mixed type machine.

#3 WCD This parameter specify a direction of compensation vector by a sign of offset value in grinding-wheel wear compensation

		Offset vale by D code	
		Minus	**Plus**
Bit 3 (WCD) of parameter No. 19607	**0**	Direction from compensation center to command end position.	Direction from command end position to compensation center
	1	Direction from command end position to compensation center	Direction from compensation center to command end position.



Direction from compensation center to command end position



Direction from command end position to compensation center

#5 CAV When an interference check finds that interference (overcutting) occurred:

0: Machining stops with the alarm PS0041, "INTERFERENCE IN CUTTER COMPENSATION".
(Interference check alarm function)

1: Machining is continued by changing the tool path to prevent interference (overcutting) from occurring. (Interference check avoidance function)

For the interference check method, see the descriptions of bit 1 (CNC) of parameter No. 5008 and bit 3 (CNV) of parameter No. 5008.

	#7	#6	#5	#4	#3	#2	#1	#0
19608		**MIR**	**PRI**			**DET**	**NI5**	

[Input type] Parameter input
[Data type] Bit path

#1 NI5 The interference check in 3-dimensional cutter compensation is performed by:
- 0: Projecting a look-ahead command position onto a plane perpendicular to the tool axis direction of a block for which compensation is planned.
  Interference avoidance function can not be used.
- 1: Projecting a look-ahead command position onto a plane that is always perpendicular to the tool axis direction. When a table rotation axis is present, checking is made in the table coordinate system.
  Interference avoidance function can be used.

#2 DET When the programming coordinate system is fastened to the table in tool center point control or 3-dimensional tool compensation, the relative position and absolute position of a specified path are:
- 0: Displayed in the programming coordinate system (fastened to the table).
- 1: Displayed in the workpiece coordinate system (not fastened to the table).

#5 PRI Among multiple end point candidates that exist when a movement is made on a rotation axis by a command such as I, J, and K when a slanted surface machining command is specified under tool center point control (type 2) or 3-dimensional tool compensation (type 2):
- 0: A combination in which the master (first rotation axis) makes a smaller angular movement is selected for a machine of tool rotation type or table rotation type. A combination in which the table (second rotation axis) makes a smaller angular movement is selected for a machine of composite type.
- 1: A combination in which the slave (second rotation axis) makes a smaller angular movement is selected for a machine of tool rotation type or table rotation type. A combination in which the tool (first rotation axis) makes a smaller angular movement is selected for a machine of composite type.

#6 MIR When programmable mirror image is applied to a linear axis in tool center point control (type 2) or 3-dimensional tool compensation (type 2), mirror image is:
- 0: Not applied to a specified I, J, or K command
- 1: Applied to a specified I, J, or K command.

19631 **Variation for determining an angle when the leading-edge offset function is performed**

[Input type] Parameter input
[Data type] Real axis
[Unit of data] degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

In the leading-edge offset function is performed, this parameter sets the range of variations applicable when the included angle of the tool direction vector (VT) and the advancing direction vector (VM) is determined to be 0°, 180°, or 90°.

For example, let the included angle between VT and VM be θ ($0 \leq \theta \leq 180$) and the angle set in this parameter be $\Delta\theta$. θ is then determined as follows:

If $0 \leq \theta \leq \Delta\theta$	$\theta = 0°$
$(180-\Delta\theta) \leq \theta \leq 180$	$\theta = 180°$
$(90-\Delta\theta) \leq \theta \leq (90+\Delta\theta)$	$\theta = 90°$

Normally, set a value around 1.0.

19632 **Distance from a programmed point (pivot point) to tool center point position (cutting point)**

[Input type] Parameter input

[Data type] Real axis
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the distance from a programmed point to the actual cutting point to perform vector calculation for 3-dimensional cutter compensation at the tool center point.

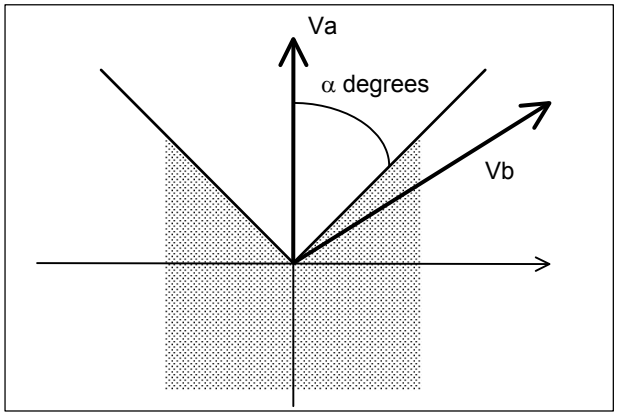
When this parameter is set to 0, the tool center point is not supported by the 3-dimensional cutter compensation function.

NOTE
This parameter can be rewritten only before the 3-dimensional cutter compensation mode is entered.

19635 **Angle used as a criterion for the interference check in 3-dimensional cutter compensation**

[Input type] Setting input
[Data type] Real axis
[Unit of data] degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
In 3-dimensional cutter compensation, if the difference in angle between two tool direction vectors is equal to or greater than the setting of this parameter, the tool direction is determined to have been changed.
When 0 is set, 45 degrees is assumed.

Let two tool direction vectors be Va and Vb. Then, if the difference in angle is α degrees or greater as shown in the figure below, the tool direction vector is determined to have changed.



19636 **Angle used to determine whether to execute the interference check/avoidance function of 3-dimensional tool compensation machining**

[Input type] Setting input
[Data type] Real path
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))

The interference check/avoidance function of 3-dimensional tool compensation machining is executed when the angle difference between the tool direction vectors for the target two points is less than the setting.

This parameter is valid when bit 1 (NI5) of parameter No. 19608 is set to 1. When the setting is 0, the angle is assumed to be 10.0 degrees.

	#7	#6	#5	#4	#3	#2	#1	#0
19640						RS3	RS2	RS1
						RS3	RS2	RS1

[Input type] Parameter input
[Data type] Bit path

 # 0 RS1
 # 1 RS2
 # 2 RS3 The tool nose rotation axis and the swivel head axis is set by the setting of the rotation axis selected by bit 0 to 2 of the parameter No.19640.

Setting of 1st rotation axis :parameter No.19681, 19682
Setting of 2nd rotation axis :parameter No.19686, 19687
Setting of 3rd rotation axis :parameter No.19691, 19692

RS3	RS2	RS1	Tool nose rotation axis	Swivel head axis
0	0	0	1st axis	2nd axis
0	0	1	1st axis	3rd axis
0	1	0	2nd axis	1st axis
0	1	1	2nd axis	3rd axis
1	0	0	3rd axis	1st axis
1	0	1	3rd axis	2nd axis
1	1	0		
1	1	1		

When (#2,#1,#0) = (1,1,0) or (1,1,1), the meaning of these parameters is the same as setting (0,0,0).(The tool nose rotation axis is the 1st rotation axis and the swivel head axis is the 2nd rotation axis.)

NOTE
The parameters No. 19691, 19692 that set the 3rd rotation axis are effective only in the Tool offset conversion function. In other 5-axis functions, these parameters are not effective.

	#7	#6	#5	#4	#3	#2	#1	#0
19641						INW	SRD	TRD
						INW	SRD	TRD

[Input type] Parameter input
[Data type] Bit path

 # 0 TRD Reference angle of the tool nose rotation axis is
0: 0 degree.
1: 180 degree.

 # 1 SRD Direction of rotation of the swivel head axis is
0: Counter-clockwise.
1: Clockwise.

 # 2 INW Amount of wear is
0: Taken into account in the offset calculate.
1: Not taken into account in the offset calculate.

19642	Reference angle of the swivel head axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] degree
[Valid data range] 0.0 to 360.0
Set the reference angle of the swivel head axis at tool measure position.

19658	Angular displacement of a rotation axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the coordinate of a rotation axis, among the rotation axes determining the tool axis direction, which is not controlled by the CNC for the tool axis direction tool length compensation function. Whether this parameter is valid or invalid is determined by the setting of bit 1 (RAP) of parameter No. 19650.

19659	Offset value for the angular displacement of a rotation axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
An offset can be applied to the angular displacement for the tool axis direction tool length compensation function to compensate for the move direction.

19660	Origin offset value of a rotation axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets an angular displacement shifted from the origin for a rotation axis for the tool axis direction tool length compensation function.

19661	Rotation center compensation vector in tool axis direction tool length compensation

[Input type] Parameter input
[Data type] Real axis

[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the vector from the first rotation axis center to the second rotation axis center for the tool axis direction tool length compensation function.

19662	Spindle center compensation vector in tool axis direction tool length compensation

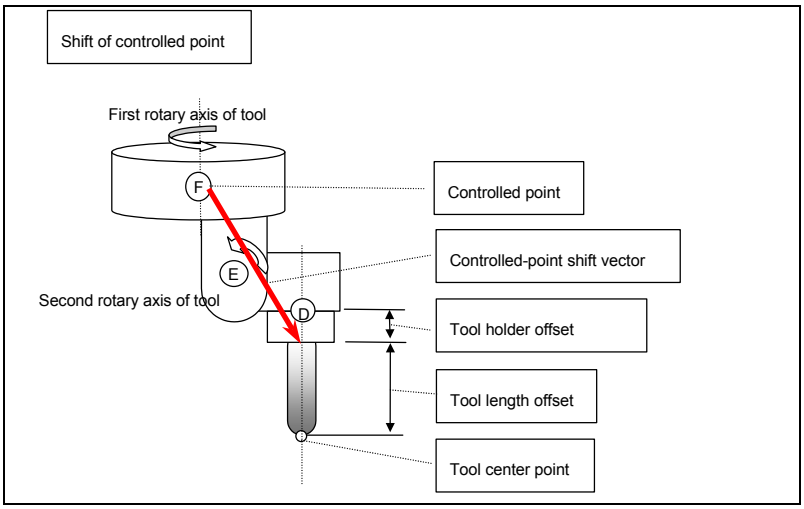
[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the compensation vector for the spindle center for the tool axis direction tool length compensation function.

	#7	#6	#5	#4	#3	#2	#1	#0
19665			SVC	SPR				

[Input type] Parameter input
[Data type] Bit path

#4 SPR The controlled point is shifted by:
0: Automatic calculation.
1: Using parameter No. 19667.

Bit 5 (SVC) of parameter No. 19665	Bit 4 (SPR) of parameter No. 19665	Shift of controlled point
0	-	Shift is not performed as not done conventionally.
1	0	The controlled point is shifted according to the result of the following automatic calculation: - (Intersection offset vector between the tool axis and the first rotation axis of the tool + intersection offset vector between the second and first rotation axes of the tool + tool holder offset (parameter No. 19666)) (See the figure below.)
1	1	The controlled point is shifted. As the shift vector, the vector set in parameter No. 19667 is used.



[Controlled-point shift vector when automatically calculated]

#5 SVC The controlled point is:
0: Not shifted.
1: Shifted.
The method of shifting is specified by bit 4 (SPR) of parameter No. 19665.

NOTE
When the machine has no rotation axis for rotating the tool (when parameter No. 19680 is set to 12 to specify the table rotation type), the controlled point is not shifted regardless of the setting of this parameter.

19666	Tool holder offset value

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set a machine-specific offset value from the control point to the tool position (tool holder offset value) for tool length compensation (in the tilted working plane mode, after G53.1), tool length compensation in tool axis direction, 3-dimensional manual handle feed, and tool center point control. For tool length compensation (except for the tilted working plane mode), however, using bit 7 (ETH) of parameter No. 19665 can specify whether to enable or disable tool holder offset.

NOTE
Set a radius value.

19667	Controlled-point shift vector

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

Set the shift vector for the controlled point. This value becomes valid when bit 5 (SVC) of parameter No. 19665 is set to 1, and bit 4 (SPR) of parameter No. 19665 is set to 1.

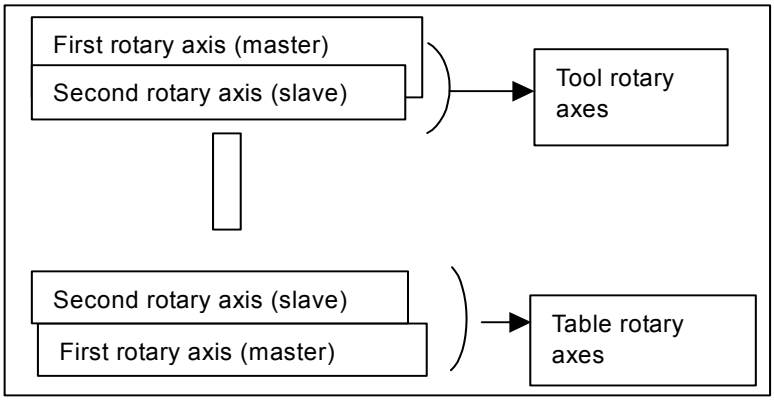
NOTE
Set a radius value.

19680	Mechanical unit type

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 21

Specify the type of the mechanical unit.

Parameter No. 19680	Mechanical unit type	Controlled rotation axis	Master and slave
0		Mechanism having no rotation axis	
2	Tool rotation type	Two rotation axes of the tool	The first rotation axis is the master, and the second rotation axis is the slave.
12	Table rotation type	Two rotation axes of the table	The first rotation axis is the master, and the second rotation axis is the slave.
21	Mixed type	One rotation axis of the tool + one rotation axis of the table	The first rotation axis is the tool rotation axis, and the second rotation axis is the table rotation axis.



NOTE
A hypothetical axis is also counted as a controlled rotary axis.
<Hypothetical axis>
In some cases, it is convenient to use an imaginary rotary axis whose angle is fixed to a certain value. For example, suppose that a tool is mounted in a tilted manner through an attachment. In such a case, the rotary axis considered hypothetically is a hypothetical axis. Bits 0 (IA1) and 1 (IA2) of parameter No. 19696 determine whether each rotary axis is an ordinary rotary axis or a hypothetical axis.

19681	Controlled-axis number for the first rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

Set the controlled-axis number for the first rotation axis.
For a hypothetical axis (when bit 0 (IA1) of parameter No. 19696 is 1), set 0.

[Example] Assuming that the axis configuration in path 1 is X,Y,Z,B,C and the axis configuration in path 2 is X,Z,C,Y,B, set the parameter to 5 in path 1 and to 3 in path 2 if C is the first rotation axis in each path.

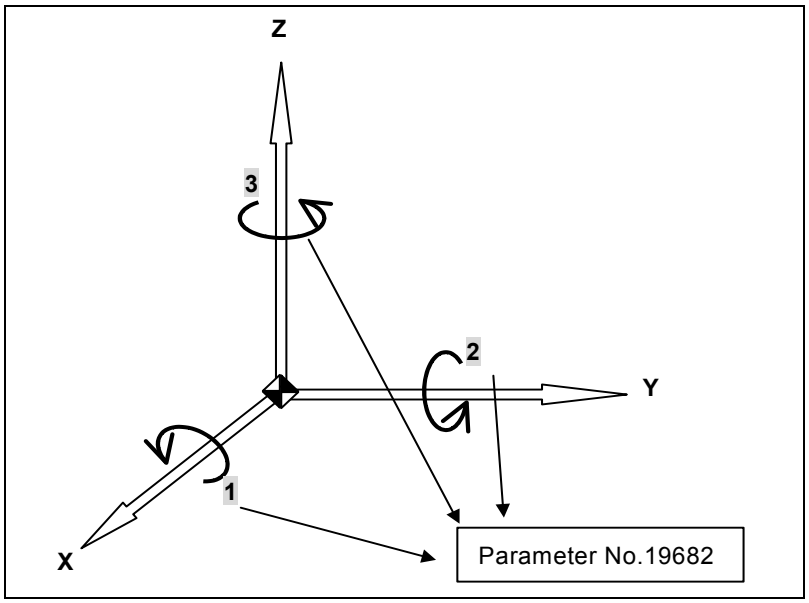
19682	Axis direction of the first rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 6

Specify the axis direction of the first rotation axis.

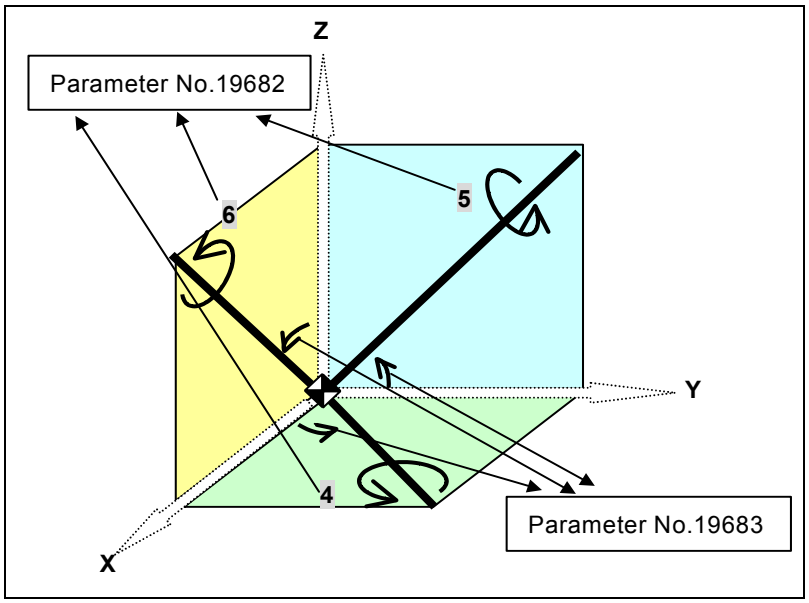
1: On X-axis
2: On Y-axis
3: On Z-axis
4: On an axis tilted a certain angle from the X-axis from the positive X-axis to positive Y-axis
5: On an axis tilted a certain angle from the Y-axis from the positive Y-axis to positive Z-axis
6: On an axis tilted a certain angle from the Z-axis from the positive Z-axis to positive X-axis

(A value 4 to 6 is to be set when the inclined rotation axis control function is used.)



19683	Inclination angle when the first rotation axis is an inclined axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] The increment system of the reference axis is to be followed.
[Valid data range] Nine digits of the least input increment (see standard parameter setting table (A).)
(-999999.999 to +999999.999 for IS-B)
When a value 1 to 3 is set in parameter No. 19682, set 0 degrees.
When a value 4 to 6 is set in parameter No. 19682, specify the inclination angle.



19684	Rotation direction of the first rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 1
Set the direction in which the first rotation axis rotates as a mechanical motion when a positive move command is issued.
0: Clockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. 19682 (right-hand thread rotation)
1: Counterclockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. 19682 (left-hand thread rotation)
Normally, 0 is set for a tool rotation axis, and 1 is set for a table rotation axis.

19685	Rotation angle when the first rotation axis is a hypothetical axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
When the first rotation axis is a hypothetical axis (bit 0 (IA1) of parameter No. 19696 is 1), set the rotation angle.

19686	Controlled-axis number for the second rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the controlled-axis number for the second rotation axis.
For a hypothetical axis (bit 1 (IA2) of parameter No. 19696 is 1), set 0.
[Example] Assuming that the axis configuration in path 1 is X,Y,Z,B,C and the axis configuration in path 2 is X,Z,C,Y,B, set the parameter to 4 in path 1 and to 5 in path 2 if B is the second rotation axis in each path.

19687	Axis direction of the second rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 6
Specify the axis direction of the second rotation axis.
1: On X-axis
2: On Y-axis
3: On Z-axis
4: On an axis tilted a certain angle from the X-axis from the positive X-axis to positive Y-axis
5: On an axis tilted a certain angle from the Y-axis from the positive Y-axis to positive Z-axis
6: On an axis tilted a certain angle from the Z-axis from the positive Z-axis to positive X-axis
(A value 4 to 6 is to be set when the inclined rotation axis control function is used.)
When the second rotation axis is the slave axis, the direction when the master axis is at 0 degrees must be set.

19688	Inclination angle when the second rotation axis is inclined

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)
If parameter No. 19687 is set to a value 1 to 3, set 0 degrees.
If parameter No. 19687 is set to a value 4 to 6, set the inclination angle.

19689	Rotation direction of the second rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 1
Set the direction in which the second rotation axis rotates as a mechanical motion when a positive move command is issued.
0: Clockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. 19687 (right-hand thread rotation)
1: Counterclockwise direction as viewed from the negative to positive direction of the axis specified in parameter No. 19687 (left-hand thread rotation)
Normally, 0 is set for a tool rotation axis, and 1 is set for a table rotation axis.

19690	Rotation angle when the second rotation axis is a hypothetical axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
When the second rotation axis is a hypothetical axis (bit 1 (IA2) of parameter No. 19696 is 1), set the rotation angle.

19691	Controlled-axis number for the third rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the controlled-axis number for the third rotation axis.
[Example] Assuming that the axis configuration in path 1 is X,Y,Z,A,B,C and the axis configuration in path 2 is X,A,Z,C,Y,B, set the parameter to 6 in path 1 and to 4 in path 2 if C is the third rotation axis in each path.

NOTE
This parameter is effective only in the Tool offset conversion function. In other 5-axis functions, this parameter is not effective.

19692	Axis direction of the third rotation axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 6
Specify the axis direction of the third rotation axis.
1: On X-axis
2: On Y-axis
3: On Z-axis
4: On X-axis tilted a certain angle from the positive direction of X-axis to the positive direction of Y-axis
5: On Y-axis tilted a certain angle from the positive direction of Y-axis to the positive direction of Z-axis
6: On Z-axis tilted a certain angle from the positive direction of Z-axis to the positive direction of X-axis
(Values 4 to 6 are to be set when the inclined rotation axis control function is used.)
When the third rotation axis is the slave axis, the direction when the master axis is at 0 degrees must be set.

NOTE
This parameter is effective only in the Tool offset conversion function. In other 5-axis functions, this parameter is not effective.

	#7	#6	#5	#4	#3	#2	#1	#0
19696		RFC	WKP		NPC		IA2	IA1

[Input type] Parameter input
[Data type] Bit path

#0 IA1 0: The first rotation axis is an ordinary rotation axis.
1: The first rotation axis is a hypothetical axis.
If IA1 is 1, set 0 as the controlled-axis number for the first rotation axis (parameter No. 19681).
Also, set parameter Nos. 19682 to 19685 on the assumption that there is a rotation axis.

#1 IA2 0: The second rotation axis is an ordinary rotation axis.
1: The second rotation axis is a hypothetical axis.
If IA2 is 1, set 0 as the controlled-axis number for the second rotation axis (parameter No. 19686).
Also, set parameter Nos. 19687 to 19690 on the assumption that there is a rotation axis.

#3 NPC In tool posture control for tool center point control (type 2), when the change of tool posture at the block end is not done with the parameters Nos. 19738 and 19739, even if the tool posture passes the singular posture,

0: Program is executed without the change of tool posture.
1: The alarm PS5421, “ILLEGAL COMMAND IN G43.4/G43.5” occurs.

#5 WKP For a 5-axis machine having a table rotation axis, as the programming coordinate system for tool center point control or 3-dimensional tool compensation machining:

0: The table coordinate system (coordinate system fixed on the rotary table) is used.
1: The workpiece coordinate system is used.

NOTE
For 3-dimensional tool compensation machining, the setting of this parameter is used only when bit 4 (TBP) of parameter No. 19746 is set to 1.

#6 RFC In tool center point control, when a command that does not move the tool center point with respect to the workpiece is issued, the feedrate of the rotation axis is:

0: The maximum cutting feedrate (parameter No. 1432).
1: A specified feedrate.

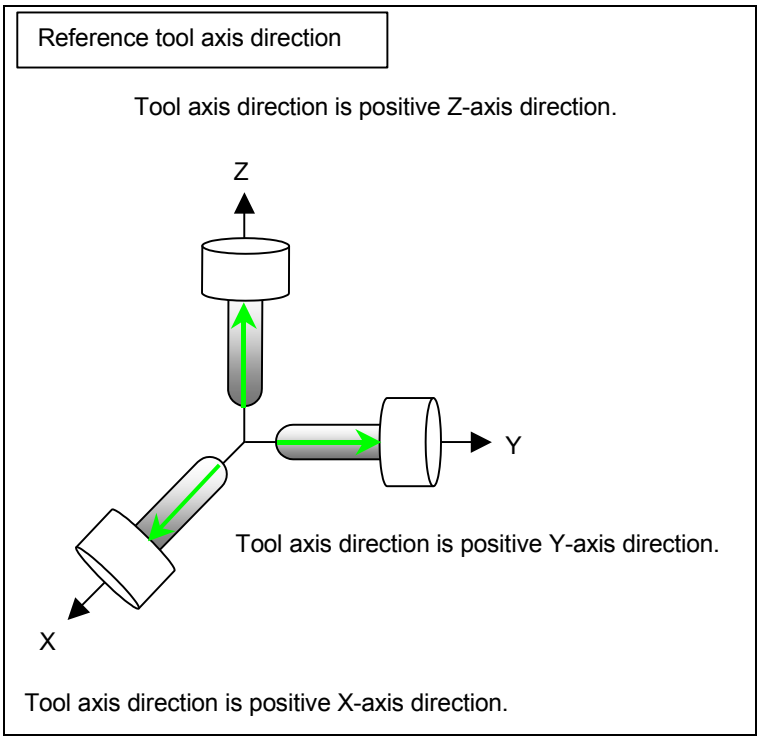
19697	Reference tool axis direction

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 3

Set the tool axis direction in the machine coordinate system when the rotation axes for controlling the tool are all at 0 degrees. Also, set the tool axis direction in the machine coordinate system in a mechanism in which only the rotation axes for controlling the table are present (there is no rotation axis for controlling the tool).

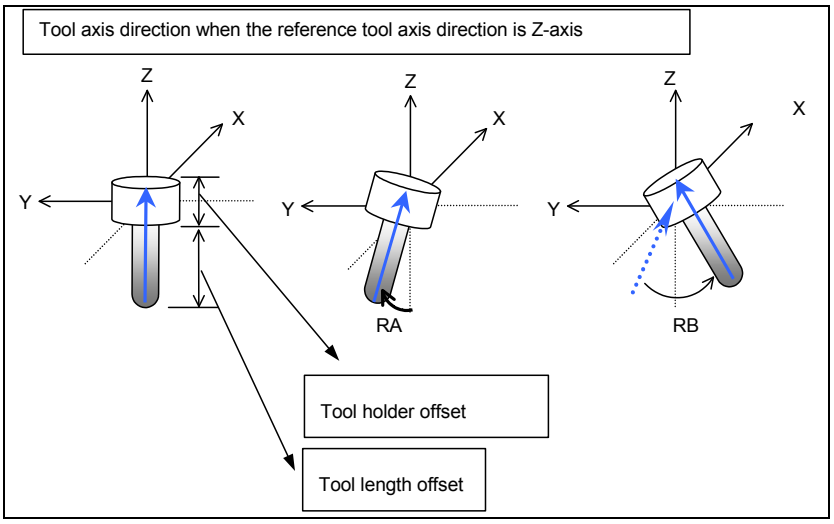
1: Positive X-axis direction
2: Positive Y-axis direction
3: Positive Z-axis direction

When the reference tool axis direction is neither the X-, Y-, nor Z-axis direction, set the reference direction in this parameter, then set appropriate angles as the reference angle RA and reference angle RB (parameter Nos. 19698 and 19699).



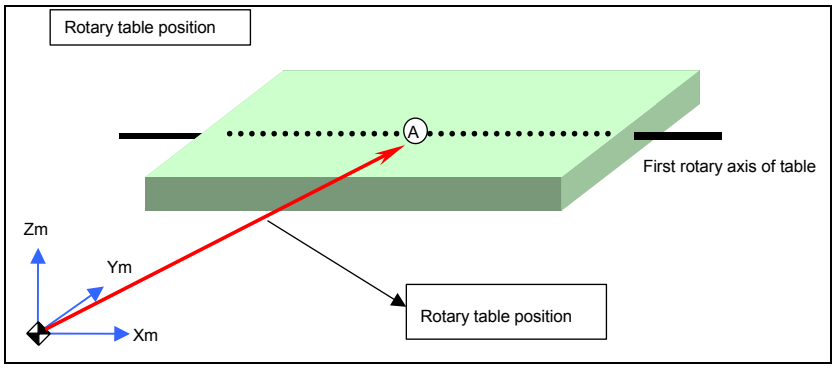
19698	Angle when the reference tool axis direction is tilted (reference angle RA)
19699	Angle when the reference tool axis direction is tilted (reference angle RB)

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)
When the reference tool axis direction (parameter No. 19697) is set to 1, the tool axis is tilted the RA degrees on the Z-axis from the positive X-axis direction to positive Y-axis direction, then the tool axis is tilted the RB degrees on the X-axis from the positive Y-axis direction to positive Z-axis direction.
When the reference tool axis direction (parameter No. 19697) is set to 2, the tool axis is tilted the RA degrees on the X-axis from the positive Y-axis direction to positive Z-axis direction, then the tool axis is tilted the RB degrees on the Y-axis from the positive Z-axis direction to positive X-axis direction.
When the reference tool axis direction (parameter No. 19697) is set to 3, the tool axis is tilted the RA degrees on the Y-axis from the positive Z-axis direction to positive X-axis direction, then the tool axis is tilted the RB degrees on the Z-axis from the positive X-axis direction to positive Y-axis direction.



19700	**Rotary table position (X-axis of the basic three axes)**
19701	**Rotary table position (Y-axis of the basic three axes)**
19702	**Rotary table position (Z-axis of the basic three axes)**

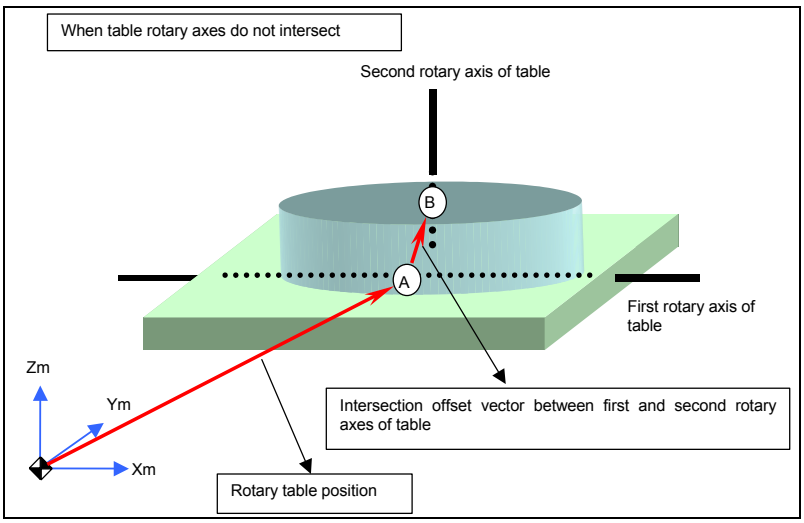
[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set these parameters when parameter No. 19680 is set to 12 or 21. The vector from the origin of the machine coordinate system to point A on the first rotation axis of the table is set as the rotary table position in the machine coordinate system.



NOTE
As point A, set a position that is easy to measure on the first rotary axis of the table.
Set a radius value.
If the rotary table is moved along the X-, Y-, or Z-axis or all of these axes, set the position of the rotary table when the machine coordinates of the X-, Y-, and Z-axes are all set to 0.

19703	Intersection offset vector between the first and second rotation axes of the table (X-axis of the basic three axes)
19704	Intersection offset vector between the first and second rotation axes of the table (Y-axis of the basic three axes)
19705	Intersection offset vector between the first and second rotation axes of the table (Z-axis of the basic three axes)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set these parameters when the first rotation axis and second rotation axis of the table do not intersect. These parameters are valid when parameter No. 19680 is set to 12. When the rotation axes for controlling the table are all at 0 degrees, the vector from point A to point B on the second rotation axis of the table is set as the intersection offset vector in the machine coordinate system.

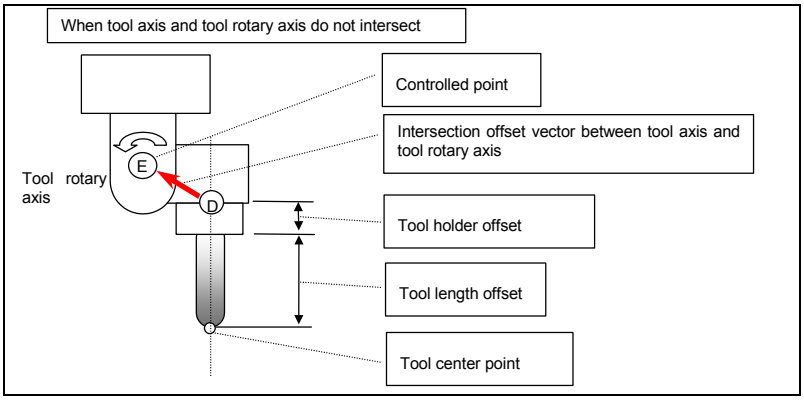


NOTE
As point B, set a position that is easy to measure on the second rotary axis of the table.
Set a radius value.

19709	Intersection offset vector between the tool axis and tool rotation axis (X-axis of the basic three axes)
19710	Intersection offset vector between the tool axis and tool rotation axis (Y-axis of the basic three axes)
19711	Intersection offset vector between the tool axis and tool rotation axis (Z-axis of the basic three axes)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set these parameters when the tool axis and tool rotation axis do not intersect.
These parameters are valid when parameter No. 19680 is set to 2 or 21.

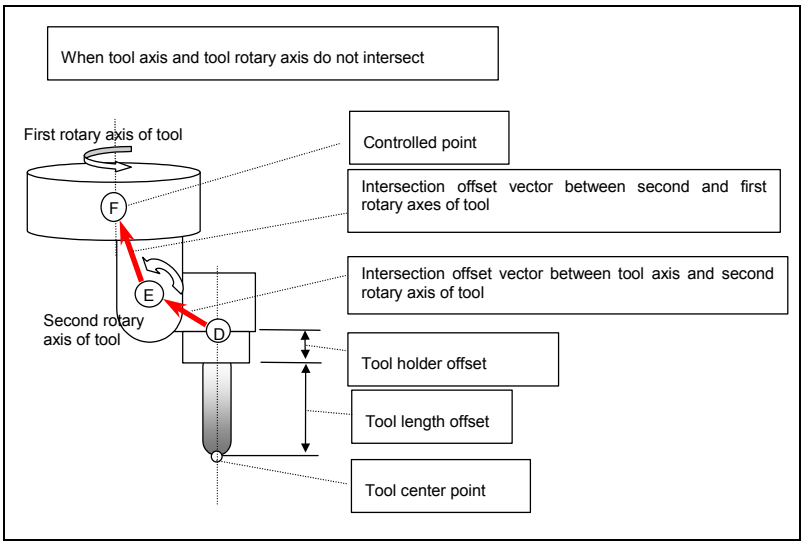
If parameter No. 19680 is 21, set the vector from point D on the tool axis to point E determined on the tool rotation axis as the intersection offset vector in the machine coordinate system when the rotation axes for controlling the tool are all at 0 degrees.
If parameter No. 19680 is 2, set the vector from point D on the tool axis to point E determined on the second rotation axis of the tool as the intersection offset vector in the machine coordinate system when the rotation axes for controlling the tool are all at 0 degrees.



NOTE
Point D is determined by adding the tool length offset and tool holder offset (parameter No. 19666) to the tool tip. As point E, set a position that is easy to measure.
Set a radius value.

19712	**Intersection offset vector between the second and first rotation axes of the tool (X-axis of the basic three axes)**
19713	**Intersection offset vector between the second and first rotation axes of the tool (Y-axis of the basic three axes)**
19714	**Intersection offset vector between the second and first rotation axes of the tool (Z-axis of the basic three axes)**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set these parameters when the rotation axes of the tool do not intersect.
These parameters are valid when parameter No. 19680 is set to 2.
Set the vector from point E on the second rotation axis of the tool to point F on the first rotation axis of the tool as the intersection offset vector in the machine coordinate system when the rotation axes for controlling the tool are all at 0 degrees.



NOTE
As point F, set a position that is easy to measure.
Set a radius value.

19738	Angle to check if Tool posture is near Singular posture or not

[Input type] Parameter input
[Data type] Real path
[Unit of data] degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table(A))
(When the increment system is IS-B, -999999.999 to +999999.999)
In the case of tool posture control for tool center point control (type 2), when the difference(angle) between a tool posture and the singular posture is less than this parameter, the tool posture is regarded as 'near the singular posture'. Then, the tool posture at the block end is changed so that the tool posture passes the singular posture during the block.
If this parameter is 0.0, the tool posture at the block end is not changed.

19739	Angle to decide that the tool posture at block end is not changed

[Input type] Parameter input
[Data type] Real path
[Unit of data] degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table(A))
(When the increment system is IS-B, -999999.999 to +999999.999)

When an appropriate value is set in parameter No. 19738 in tool posture control for tool center point control (type 2), a tool posture near the singular point may occur during the execution of a block. If this happens, change the tool posture at the end point so that the singular posture is passed within the block. With respect to the angle of the rotation axis nearer to the workpiece before and after the tool posture is changed (the rotation axis is the master axis when the tool turns, the slave axis when the table turns, or the table rotation axis when the rotation is of the mixed type), however, the tool posture must not be changed if both the difference between the angle after the change and (angle before the change + 180 degrees) and the difference between the angle after the change and (angle before the change − 180 degrees) are equal to or greater than the value set in the parameter.

19741	Upper limit of the movement range of the first rotation axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the upper limit of the movement range of the first rotation axis in tool center point control (type 2), 3-dimensional cutter compensation (type 2), or tool axis direction control of the tilted working plane command (G53.1). When the movement range of the first rotation axis is not specified, this parameter and parameter No. 19742 must both be set to 0.

19742	Lower limit of the movement range of the first rotation axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the lower limit of the movement range of the first rotation axis in tool center point control (type 2), 3-dimensional cutter compensation (type 2), or tool axis direction control of the tilted working plane command (G53.1). When the movement range of the first rotation axis is not specified, this parameter and parameter No. 19741 must both be set to 0.

19743	Upper limit of the movement range of the second rotation axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the upper limit of the movement range of the second rotation axis in tool center point control (type 2), 3-dimensional cutter compensation (type 2), or tool axis direction control of the tilted working plane command (G53.1). When the movement range of the second rotation axis is not specified, this parameter and parameter No. 19744 must both be set to 0.

19744	Lower limit of the movement range of the second rotation axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the lower limit of the movement range of the second rotation axis in tool center point control (type 2), 3-dimensional cutter compensation (type 2), or tool axis direction control of the tilted working plane command (G53.1). When the movement range of the second rotation axis is not specified, this parameter and parameter No. 19743 must both be set to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
19746		CRS		TBP	LOZ	LOD	PTD	

[Input type] Parameter input
[Data type] Bit path

#1 PTD When 3-dimensional cutter compensation is performed for a table rotation type machine, the direction of the tool is:
0: Specified by parameter Nos. 19697, 19698, and 19699.
1: Specified as a direction perpendicular to the plane specified by G17, G18, or G19.

#2 LOD As the tool length for 3-dimensional machining manual feed:
0: The value of parameter No. 12318 is used.
1: The tool length currently used for tool length compensation is used.

#3 LOZ When bit 2 (LOD) of parameter No. 19746 is set to 1 and tool length compensation is not applied, as the tool length for 3-dimensional machining manual feed:
0: The value of parameter No. 12318 is used.
1: 0 is used.

#4 TBP For a 5-axis machine having a table rotation axis, as the programming coordinate system for 3-dimensional tool compensation machining:
0: The workpiece coordinate system is used.
1: The setting of bit 5 (WKP) of parameter No. 19696 is used.

#6 CRS In tool center point control, when the deviation from the path during movement at the specified cutting feedrate or rapid traverse rate is determined to exceed the limit:
0: The feedrate or rapid traverse rate is not decreased.
1: The feedrate or rapid traverse rate is controlled so that the limit of the deviation from the path set in the parameter for the cutting feed or rapid traverse is not exceeded.
When this parameter is set to 1:
In the rapid traverse mode, the rapid traverse rate is decreased so that the deviation from the path does not exceed the limit specified in parameter No. 19751.
In the cutting feed mode, the cutting feedrate is decreased so that the deviation from the path does not exceed the limit specified in parameter No. 19752.

19751	Limit of the deviation from the path (for rapid traverse)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)

[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the limit of the deviation from the path in the rapid traverse mode in tool center point control.
If the tool moves at the specified rate, the deviation from the path may exceed the value specified in this parameter. In this case, the rate is decreased so that the tool moves along the path.
This parameter is valid when bit 6 (CRS) of parameter No. 19746 is set to 1.
When 0 is set, the least input increment is assumed to be the limit of the deviation from the path.
If a negative value is set, the rapid traverse rate is not decreased.

NOTE
The error generated after the rate is decreased may be smaller than the value set in this parameter depending on the calculation error.

19752	Limit of the deviation from the path (for cutting feed)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the limit of the deviation from the path in the cutting feed mode in tool center point control.
If the tool moves at the specified rate, the deviation from the path may exceed the value specified in this parameter. In this case, the rate is decreased so that the tool moves along the path.
This parameter is valid when bit 6 (CRS) of parameter No. 19746 is set to 1.
When 0 is set, the least input increment is assumed to be the limit of the deviation from the path.
If a negative value is set, the cutting feedrate is not decreased.

NOTE
The error generated after the rate is decreased may be smaller than the value set in this parameter depending on the calculation error.

	#7	#6	#5	#4	#3	#2	#1	#0
19754	SPM		INZ					

[Input type] Parameter input
[Data type] Bit axis

#5 INZ If, in tool center point control and 3-dimensional cutter compensation, a table coordinate system command is issued,
0: In the state in which each function is started, the workpiece coordinate system is fixed to the rotary table, and becomes a table coordinate system.
1: Regardless of the table rotation axis position when each function is started, the workpiece table is fixed to the rotary table, with the table rotation axis position being 0, and becomes a table coordinate system.

#7 SPM The rotation axis position used as the reference when the parameters related to the functions below, parameters Nos. 19681 to 19714, are set is:

0: Absolute coordinates.
1: Machine coordinates

This parameter is effective to the functions below.

- Tool center point control
- Smooth TCP
- Tool posture control
- 3-dimensional tool compensation
- Cutting point command
- Workpiece setting error compensation
- Tilted working plane command
- Tilted working plane command with guidance

24000	**ATR value corresponding to slave 01 on first FSSB line**
24001	**ATR value corresponding to slave 02 on first FSSB line**
to	**to**
24031	**ATR value corresponding to slave 32 on first FSSB line**

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1001 to 1046, 2001 to 2016, 3001 to 3004, 1000

Each of these parameters sets the value (ATR value) of the address translation table corresponding to each of slave 1 to slave 32 on first FSSB line (first optical connector).
The slave is a generic term for servo amplifiers, spindle amplifiers and separate detector interface units connected via an FSSB optical cable to the CNC. Numbers 1 to 32 are assigned to slaves, with younger numbers sequentially assigned to slaves closer to the CNC.
A 2-axis amplifier consists of two slaves, and a 3-axis amplifier consists of three slaves. In each of these parameters, set a value as described below, depending on whether the slave is an amplifier, separate detector interface unit, or nonexistent.

- When the slave is a servo amplifier:
  Set the axis number of a servo amplifier to allocate (value set with parameter No. 1023) plus 1000.
- When the slave is a spindle amplifier:
  Set the spindle number of a spindle to allocate (value set with parameter No. 3717) plus 2000.
- When the slave is a separate detector interface unit:
  Set 3001, 3002, 3003, and 3004, respectively, for the first (one connected nearest to the CNC), second, third, and fourth separate detector interface units.
- When the slave is nonexistent:
  Set 1000.

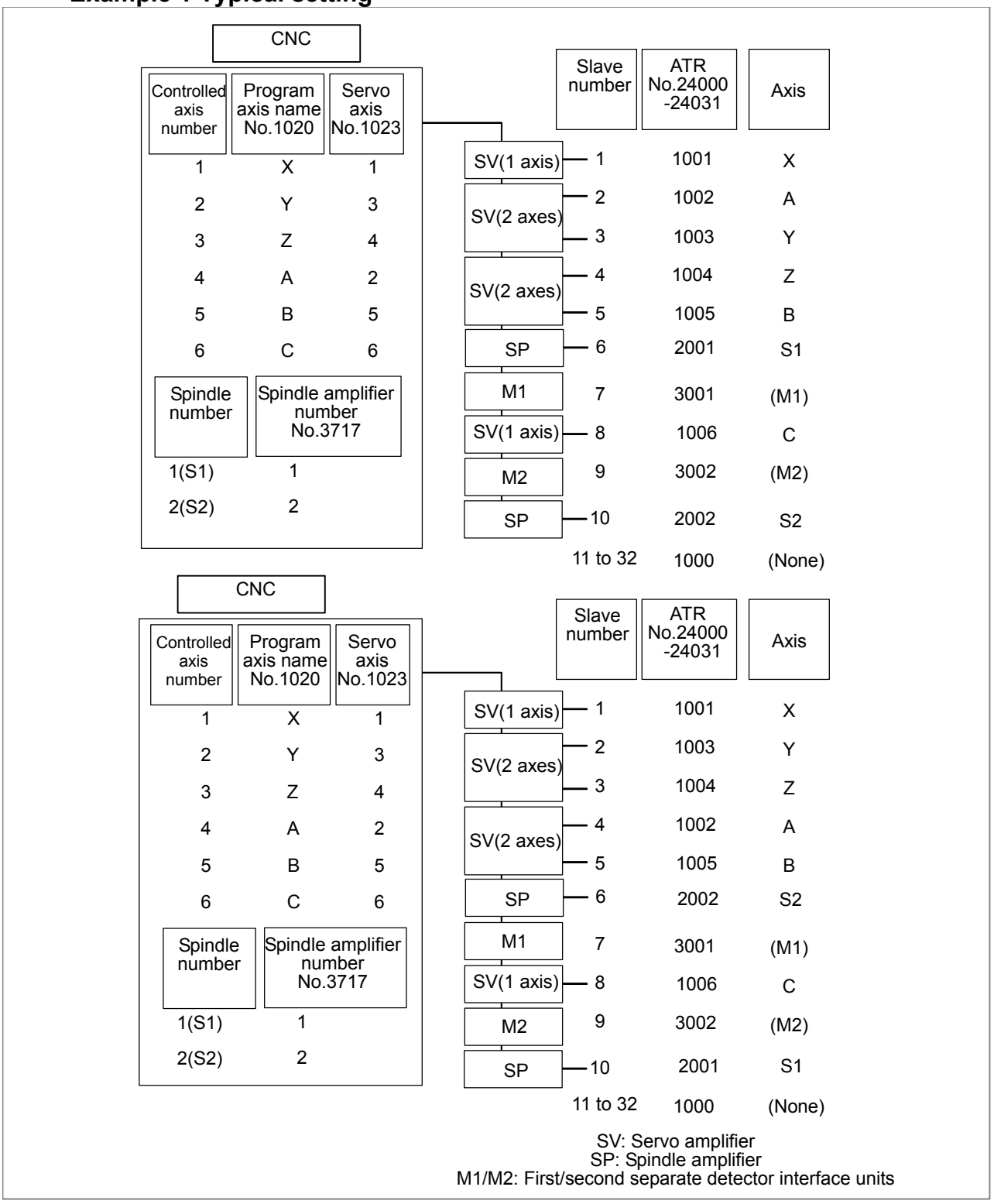
NOTE

1 When the electronic gear box (EGB) function is used
Although an amplifier is not actually required for an EGB dummy axis, set this parameter with assuming that a dummy amplifier is connected. To put it another way, specify this parameter with a value set in the EGB dummy axis parameter (No. 1023) plus 1000, instead of “1000”, as an address translation table value for one of non-existent slaves.

2 When the FSSB is set to the automatic setting mode (when the bit 0 (FMD) of parameter No. 1902 is set to 0), parameter Nos. 24000 to 24031 are automatically set as data is input on the FSSB setting screen. When the manual setting 2 mode is set (when the bit 0 (FMD) of parameter No. 1902 is set to 1), be sure to directly set values in parameter Nos. 24000 to 24031.

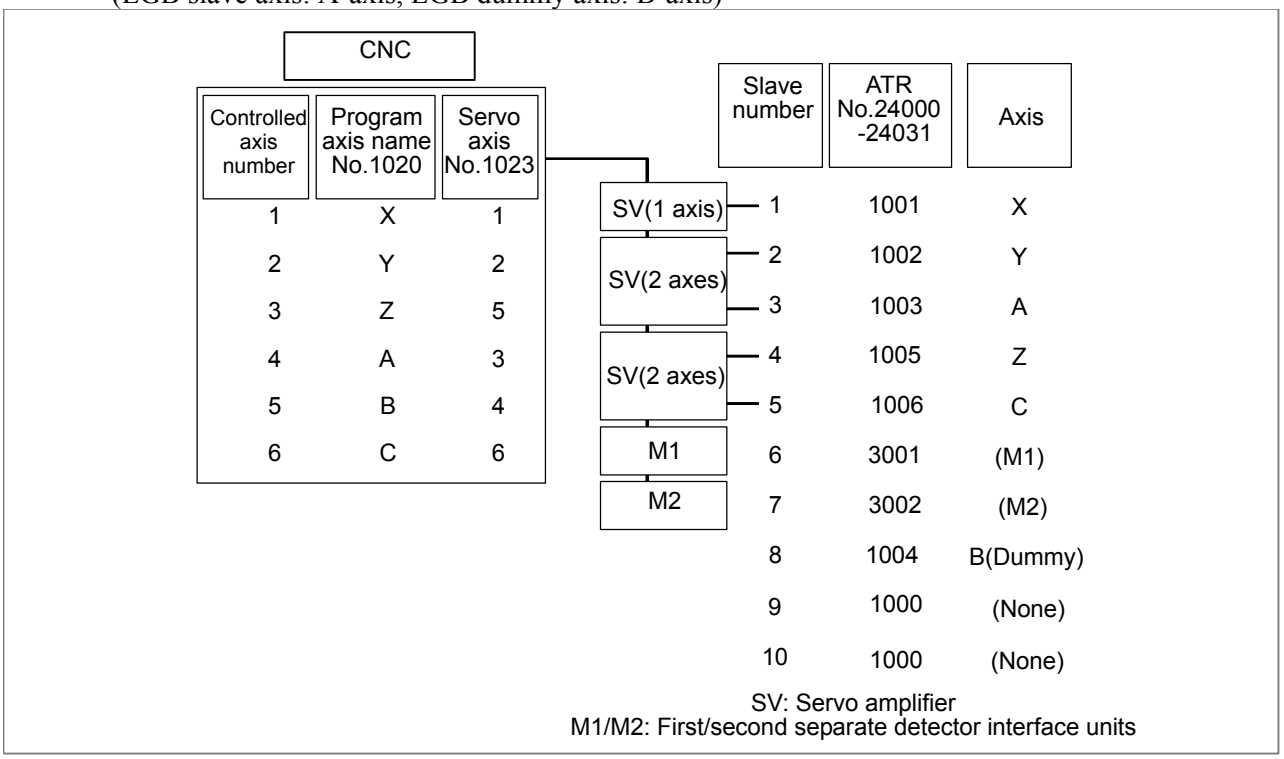
Example of axis configuration and parameter settings

- Example 1 Typical setting



- Example 2 Setting with a dummy axis in use

Example of axis configuration and parameter settings when the electronic gear box (EGB) function is used
(EGB slave axis: A-axis, EGB dummy axis: B-axis)



24032	**ATR value corresponding to slave 01 on second FSSB line**
24033	**ATR value corresponding to slave 02 on second FSSB line**
to	**to**
24063	**ATR value corresponding to slave 32 on second FSSB line**

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1001 to 1046, 2001 to 2016, 3005 to 3008, 1000

Each of these parameters sets the value (ATR value) of the address translation table corresponding to each of slave 1 to slave 32 on second FSSB line (second optical connector). Set these parameters only when a servo axis control card with two optical connectors (FSSB lines) is used.

To specify these parameters, follow the same procedure as for the first FSSB line (parameters Nos. 24000 to 24031). Note, however, that the valid data range varies depending on the separate detector interface unit used.

- When the slave is a separate detector interface unit:
  Set 3005, 3006, 3007, and 3008, respectively, for the first (one connected nearest to the CNC), second, third, and fourth separate detector interface units.

24064	ATR value corresponding to slave 01 on third FSSB line
24065	ATR value corresponding to slave 02 on third FSSB line
to	to
24095	ATR value corresponding to slave 32 on third FSSB line

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word
[Valid data range] 1049 to 1078, 2001 to 2016, 3009 to 3012, 1000

Each of these parameters sets the value (ATR value) of the address translation table corresponding to each of slave 1 to slave 32 on third FSSB line. Set these parameters only when an additional axis board is used.

To specify these parameters, follow the same procedure as for the first FSSB line (parameters Nos. 24000 to 24031). Note, however, that the valid data range varies.

- When the slave is a separate detector interface unit:
  Set 3009, 3010, 3011, and 3012, respectively, for the first (one connected nearest to the CNC), second, third, and fourth separate detector interface units.

24096	Connector number for the first or ninth separate detector interface unit
24097	Connector number for the second or tenth separate detector interface unit
24098	Connector number for the third or eleventh separate detector interface unit
24099	Connector number for the fourth or twelfth separate detector interface unit
24100	Connector number for the fifth separate detector interface unit
24101	Connector number for the sixth separate detector interface unit
24102	Connector number for the seventh separate detector interface unit
24103	Connector number for the eighth separate detector interface unit

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 8

Set a connector number for the connector to which a separate detector interface unit is attached if the separate detector interface unit is to be used. The following table lists the necessary settings. Be sure to specify 0 for connectors not in use.

Correspondence between connectors and connector numbers	
Connector	**Connector number**
JF101	1
JF102	2
JF103	3

Correspondence between connectors and connector numbers	
Connector	**Connector number**
JF104	4
JF105	5
JF106	6
JF107	7
JF108	8

(Setting example)

Controlled axis	Connector to which each separate detector interface unit is attached				Parameter setting			
	1st connector	2nd connector	5th connector	6th connector	No. 24096	No. 24097	No. 24100	No. 24101
X1	JF101	—	—	—	1	0	0	0
Y1	—	JF102	—	—	0	2	0	0
Z1	—	—	JF102	—	0	0	2	0
X2	—	JF101	—	—	0	1	0	0
Y2	—	—	—	JF101	0	0	0	1
Z2	—	—	—	—	0	0	0	0
A1	—	—	JF101	—	0	0	1	0
B1	—	—	—	JF102	0	0	0	2
C1	—	JF104	—	—	0	4	0	0
A2	JF102	—	—	—	2	0	0	0
B2	—	JF103	—	—	0	3	0	0
C2	—	—	—	JF103	0	0	0	3

NOTE
1 Specify these parameters when separate detector interface units are used.
2 Parameters Nos. 24096 to 24103 are specified automatically when data is entered on the FSSB setting screen if the FSSB setting mode in use is the automatic setting mode (bit 0 (FMD) of parameter No. 1902 = “0”). If the manual setting 2 mode (bit 0 (FMD) of parameter No. 1902) = “1”), specify the parameters directly.

	#7	#6	#5	#4	#3	#2	#1	#0
25650								**SSO**

[Input type] Parameter input
[Data type] Bit

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 SSO High precision oscillation function is
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
25651					**HST**	**SGS**	**FFS**	**OST**

[Input type] Parameter input

[Data type] Bit path

#0 OST During oscillation motion, if G80 command or reset is specified,
0: Oscillation axis moves to point R and stops. (Standard specification)
1: Oscillation axis decelerates and stops.

#1 FFS When oscillation motion is started by G81.1 command,
0: Oscillation motion is begun, after the oscillation axis passes point R and the center point between upper and lower dead points are passed. (Standard specification)
1: Movements to point R and to the center point between upper and lower dead points is omitted, and oscillation motion can be immediately started.
If this parameter is set to 1, the first dead point is as follows.

State before G81.1 command	The first dead point
First command of oscillation motion after the power is turned on or in the state of reset.	Lower dead point
In the state that command of cancel was specified on the way to lower dead point at last oscillation motion.	
In the state that command of cancel was specified on the way to upper dead point at last oscillation motion.	Upper dead point

NOTE
If this parameter is effective, parameter OST(No,25651#0) need be set to 1.

#2 SGS If oscillation motion is canceled by oscillation start signal CHPST <Gn051.6>,
0: Oscillation axis moves to point R and stops. (Standard specification)
1: Oscillation axis decelerates and stops.
If the oscillation axis decelerates and stops by oscillation start signal CHPST, use each axis workpiece coordinate system preset signal WPRST1-WPRST8<Gn358> together.
Refer to the following manual for details of the setting method and so on.
"FANUC SERIES 30i/31i/32i-MODEL B CONNECTION MANUAL (FUNCTION)" (B-64483EN-1) "Each Axis Workpiece Coordinate System Preset Signal"

NOTE
If this parameter is effective, parameter OST(No,25651#0) need be set to 1.

#3 HST During oscillation motion, if oscillation hold signal *CHLD <Gn051.7> is set to"0" from "1",
0: Oscillation axis moves to point R and suspends. (Standard specification)
1: Oscillation axis decelerates and suspends.

NOTE
If this parameter is effective, parameter OST and SGS (No.25651#0 and #2) need be set to 1.

25652	Maximum allowable acceleration rate of oscillation motion

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)

[Valid data range] Refer to standard parameter setting table (D).
Set a maximum allowable acceleration rate for oscillation axis during oscillation motion (with sine curve feedrate).

If a value greater than maximum value is set, the value is clamped to maximum value.
If 0 is set, maximum value is assumed to be set.

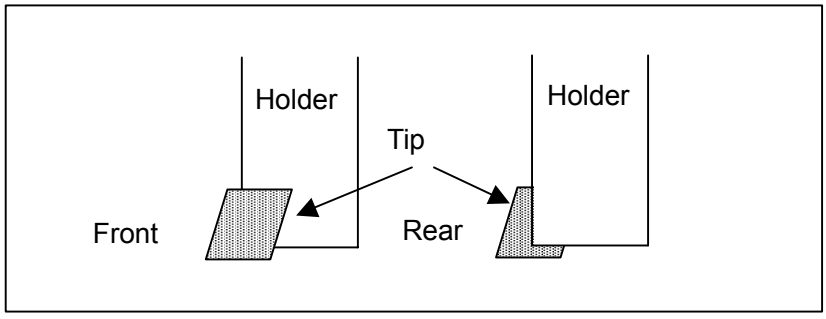
25653	Acceleration rate of starting or cancellation oscillation motion

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec^2, inch/sec^2, deg/sec^2 (machine unit)
[Valid data range] Refer to standard parameter setting table (D).
It is the acceleration rate of starting motion (from point R to center point between upper and lower dead points) or cancel motion.
Moreover, when override is changed during oscillation motion, the acceleration/ deceleration is set by this parameter.
If a value greater than maximum value is set, the value is clamped to maximum value.
If 0 is set, maximum value is assumed to be set.

	#7	#6	#5	#4	#3	#2	#1	#0
27350								GTP

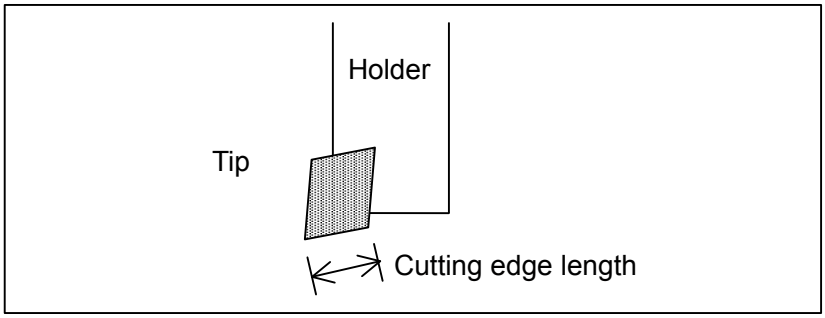
[Input type] Parameter input
[Data type] Bit

#0 GTP When a general-purpose tool is drawn in animated simulation, the tip is:
0: Positioned on the front.
1: Positioned on the rear.



27351	Cutting edge length applied when a general-purpose tool is drawn in animated simulation

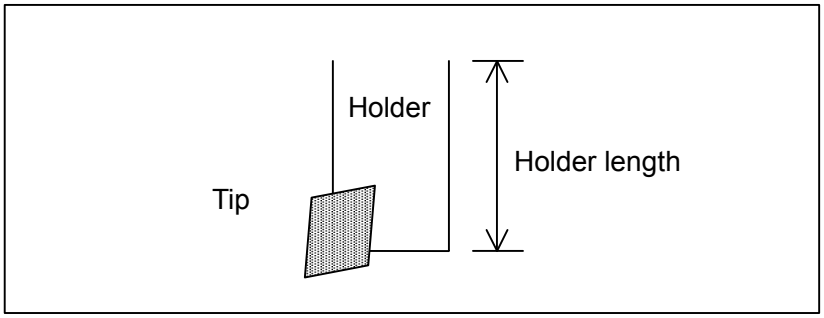
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the cutting edge length applied when a general-purpose tool is drawn in animated simulation.



When 0 is set, 12 mm for metric input or 0.4724 inch for inch input is assumed.

27352	Holder length applied when a general-purpose tool is drawn in animated simulation

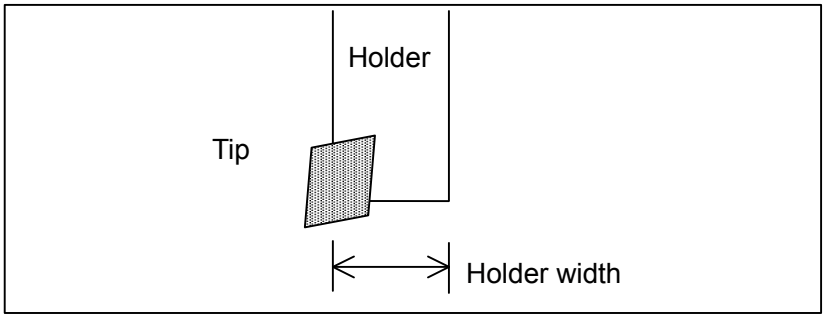
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the holder length applied when a general-purpose tool is drawn in animated simulation.



When 0 is set, 50 mm for metric input or 1.9685 inch for inch input is assumed.

27353	Holder width applied when a general-purpose tool is drawn in animated simulation

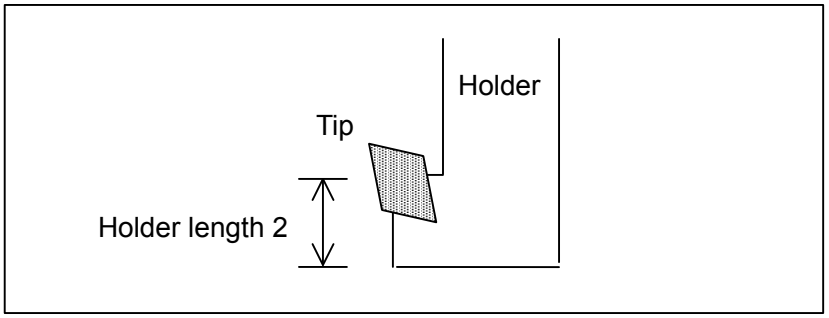
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the holder width applied when a general-purpose tool is drawn in animated simulation.



When 0 is set, 14 mm for metric input or 0.5512 inch for inch input is assumed.

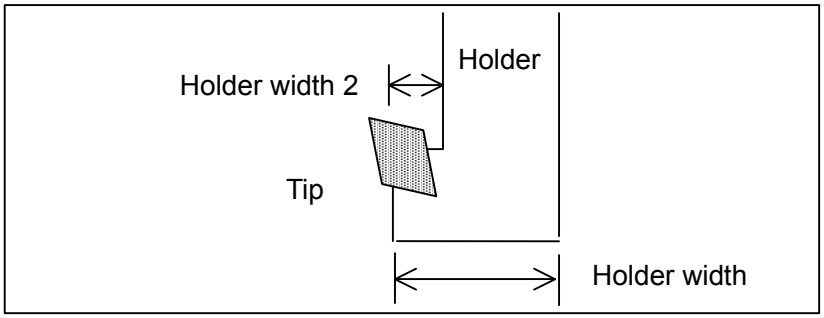
27354	Holder length 2 applied when a general-purpose tool is drawn in animated simulation

[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the holder length 2 applied when a general-purpose tool is drawn in animated simulation.



27355	Holder width 2 applied when a general-purpose tool is drawn in animated simulation

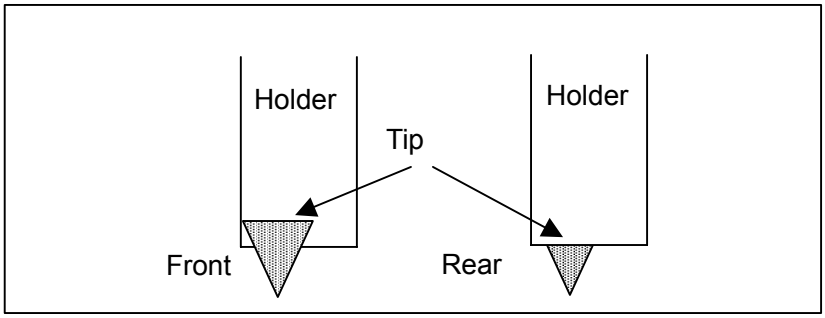
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the holder width 2 applied when a general-purpose tool is drawn in animated simulation.



	#7	#6	#5	#4	#3	#2	#1	#0
27356								TTP

[Input type] Parameter input
[Data type] Bit

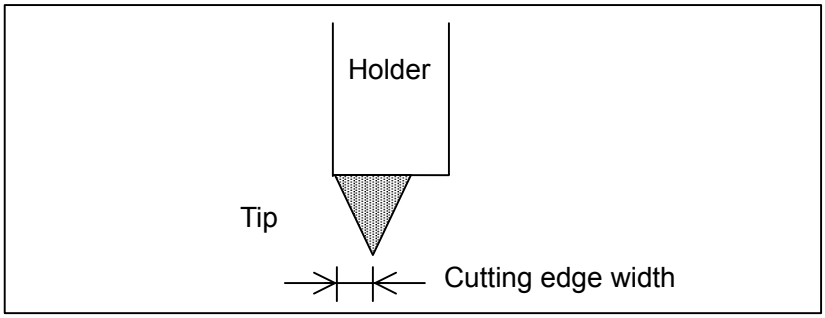
#0 TTP When a threading tool is drawn in animated simulation, the tip is:
0: Positioned on the front.
1: Positioned on the rear.



27357	Cutting edge width applied when a threading tool is drawn in animated simulation

[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger

This parameter sets the cutting edge width applied when a threading tool is drawn in animated simulation.

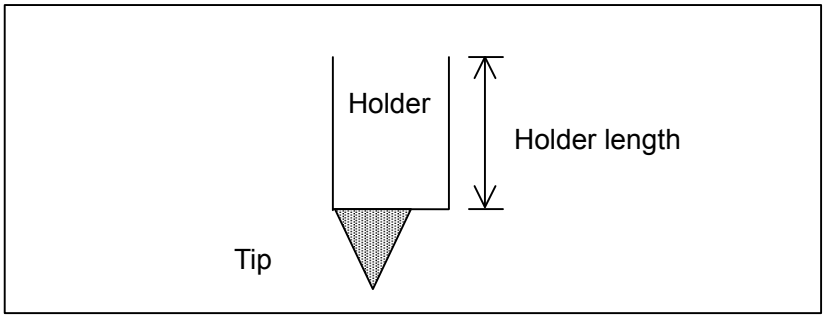


When 0 is set, 3 mm for metric input or 0.11811 inch for inch input is assumed.

27358	Holder length applied when a threading tool is drawn in animated simulation

[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger

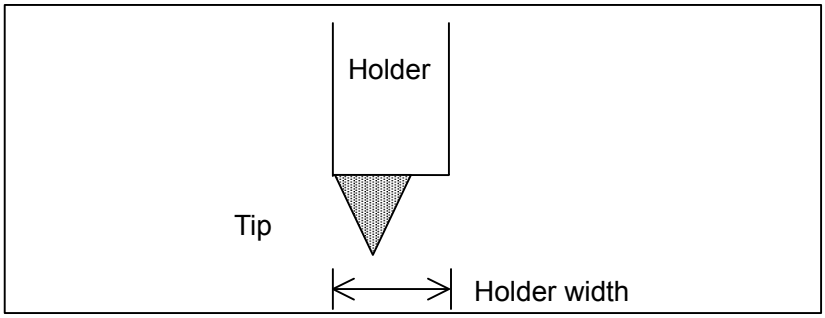
This parameter sets the holder length applied when a threading tool is drawn in animated simulation.



When 0 is set, 50 mm for metric input or 1.9685 inch for inch input is assumed.

27359	Holder width applied when a threading tool is drawn in animated simulation

[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the holder width applied when a threading tool is drawn in animated simulation.

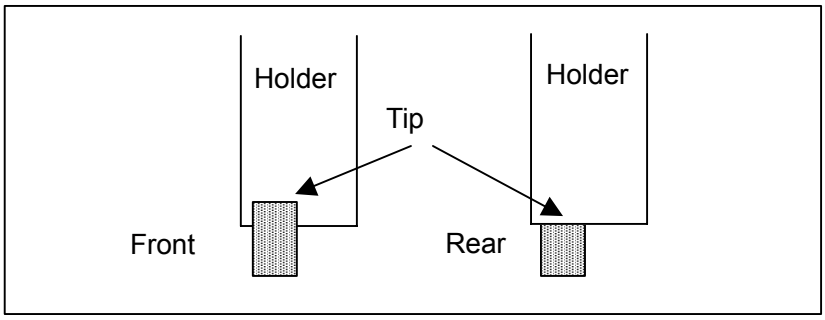


When 0 is set, 14 mm for metric input or 0.5512 inch for inch input is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
27360								GVP

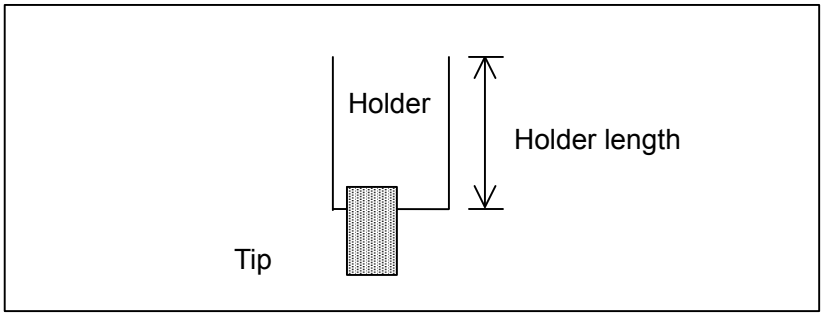
[Input type] Parameter input
[Data type] Bit

#0 GVP When a groove cutting tool is drawn in animated simulation, the tip is:
0: Positioned on the front.
1: Positioned on the rear.



27361	Holder length applied when a groove cutting tool is drawn in animated simulation

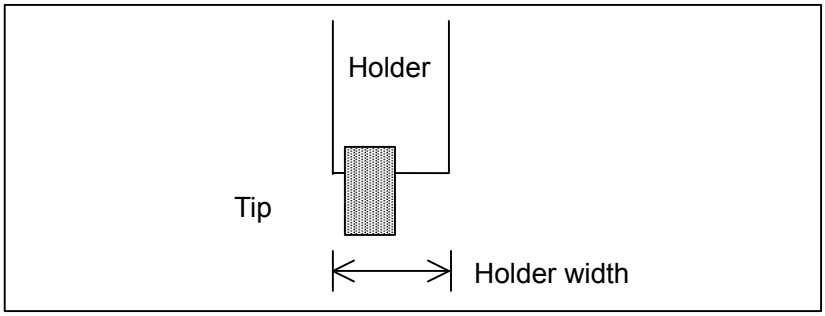
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the holder length applied when a groove cutting tool is drawn in animated simulation.



When 0 is set, 50 mm for metric input or 1.9685 inch for inch input is assumed.

27362	Holder width applied when a groove cutting tool is drawn in animated simulation

[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the holder width applied when a groove cutting tool is drawn in animated simulation.

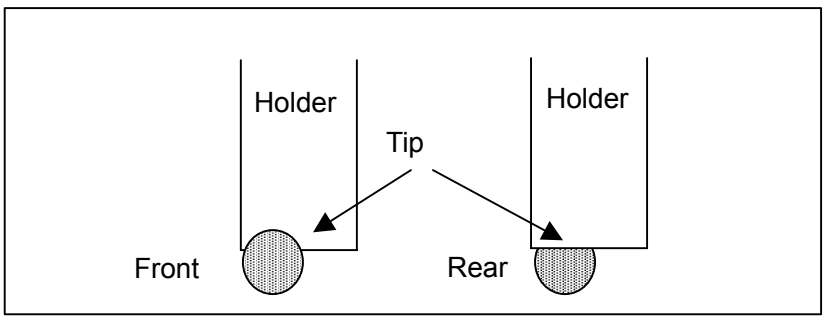


When 0 is set, 14 mm for metric input or 0.5512 inch for inch input is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
27363								BTP

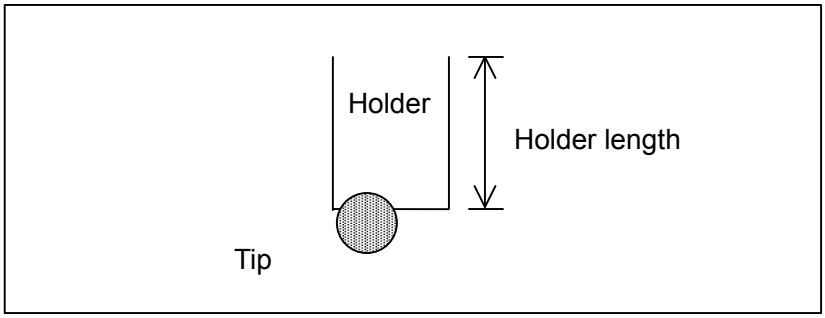
[Input type] Parameter input
[Data type] Bit

#0 BTP When a round-nose tool is drawn in animated simulation, the tip is:
0: Positioned on the front.
1: Positioned on the rear.



27364	Holder length applied when a round-nose tool is drawn in animated simulation

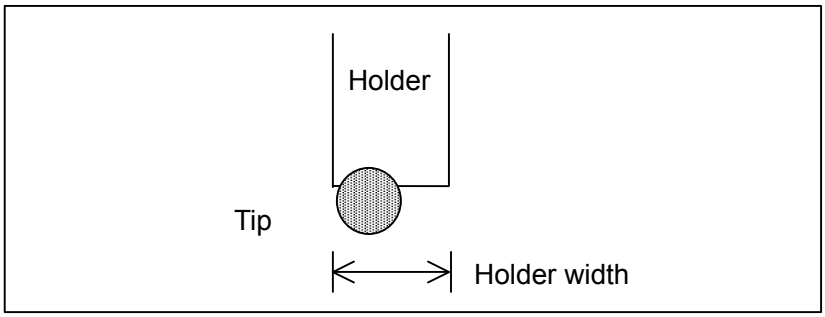
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the holder length applied when a round-nose tool is drawn in animated simulation.



When 0 is set, 50 mm for metric input or 1.9685 inch for inch input is assumed.

27365	Holder width applied when a round-nose tool is drawn in animated simulation

[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the holder width applied when a round-nose tool is drawn in animated simulation.

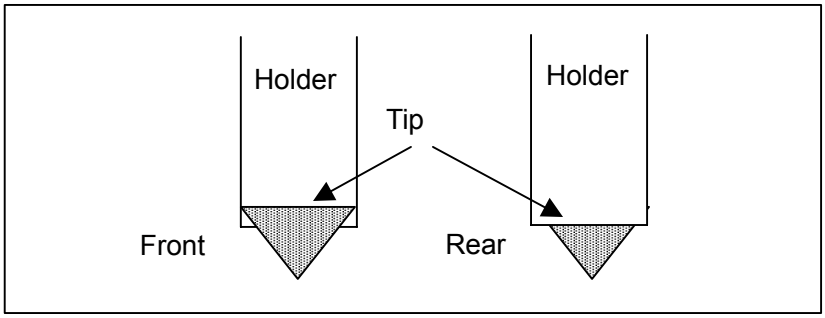


When 0 is set, 14 mm for metric input or 0.5512 inch for inch input is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
27366								STP

[Input type] Parameter input
[Data type] Bit

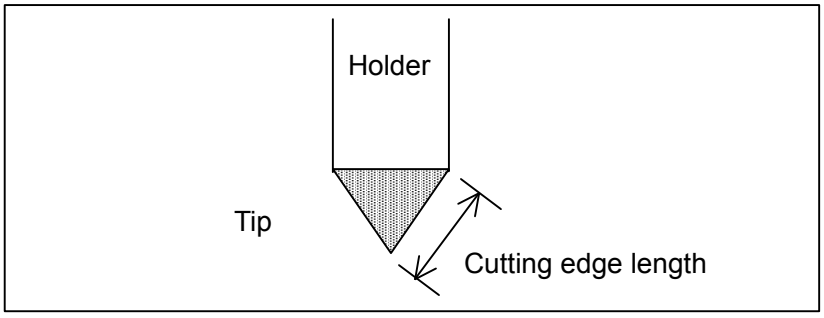
#0 STP When a point nose straight tool is drawn in animated simulation, the tip is:
0: Positioned on the front.
1: Positioned on the rear.



27367	Cutting edge length applied when a point nose straight tool is drawn in animated simulation

[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger

This parameter sets the cutting edge length applied when a point nose straight tool is drawn in animated simulation.

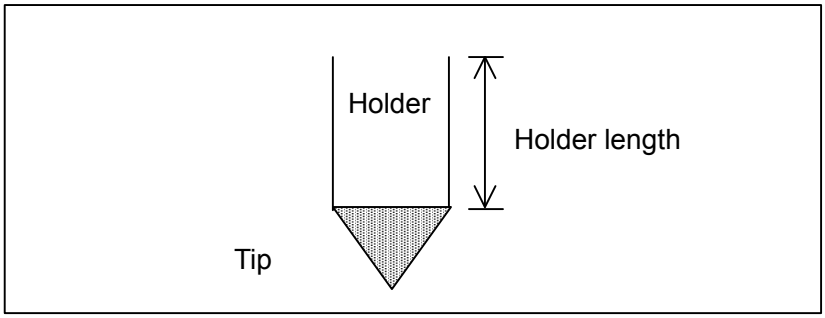


When 0 is set, 12 mm for metric input or 0.4724 inch for inch input is assumed.

27368	Holder length applied when a point nose straight tool is drawn in animated simulation

[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger

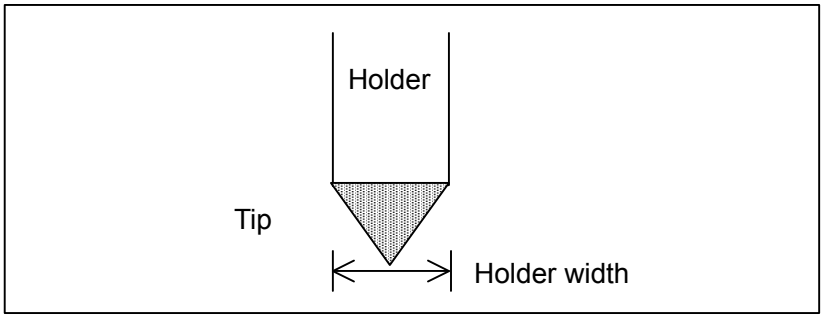
This parameter sets the holder length applied when a point nose straight tool is drawn in animated simulation.



When 0 is set, 50 mm for metric input or 1.9685 inch for inch input is assumed.

27369	Holder width applied when a point nose straight tool is drawn in animated simulation

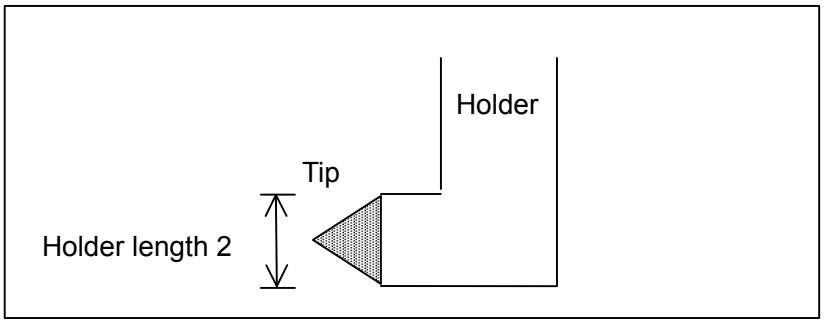
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the holder width applied when a point nose straight tool is drawn in animated simulation.



When 0 is set, 14 mm for metric input or 0.5512 inch for inch input is assumed.

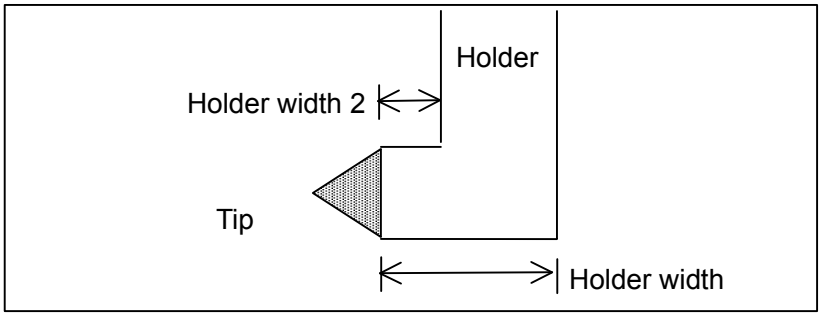
27370	Holder length 2 applied when a point nose straight tool is drawn in animated simulation

[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the holder length 2 applied when a point nose straight tool is drawn in animated simulation.



27371	Holder width 2 applied when a point nose straight tool is drawn in animated simulation

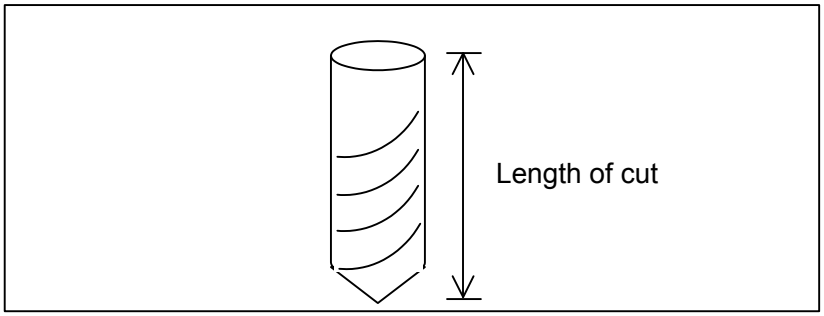
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the holder width 2 applied when a point nose straight tool is drawn in animated simulation.



27372	Length of cut applied when a drill tool is drawn in animated simulation

[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger

This parameter sets the length of cut applied when a drill tool is drawn in animated simulation.

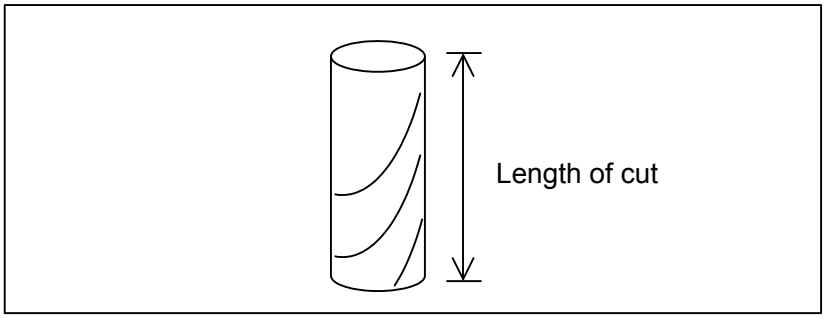


When 0 is set, 50 mm for metric input or 1.9685 inch for inch input is assumed.

27373	Length of cut applied when a flat end milling cutter is drawn in animated simulation

[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger

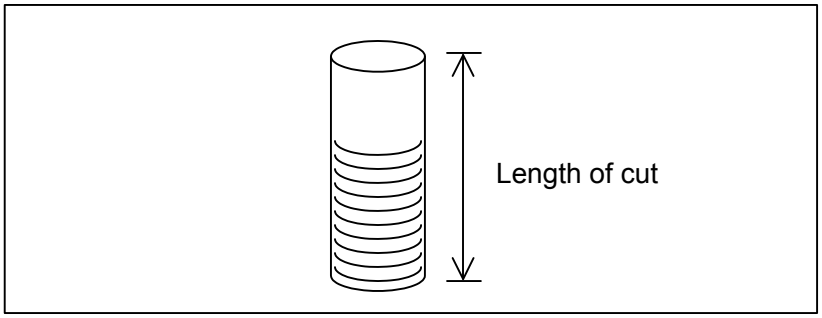
This parameter sets the length of cut applied when a flat end milling cutter is drawn in animated simulation.



When 0 is set, 50 mm for metric input or 1.9685 inch for inch input is assumed.

27374	Length of cut applied when a tapping tool is drawn in animated simulation

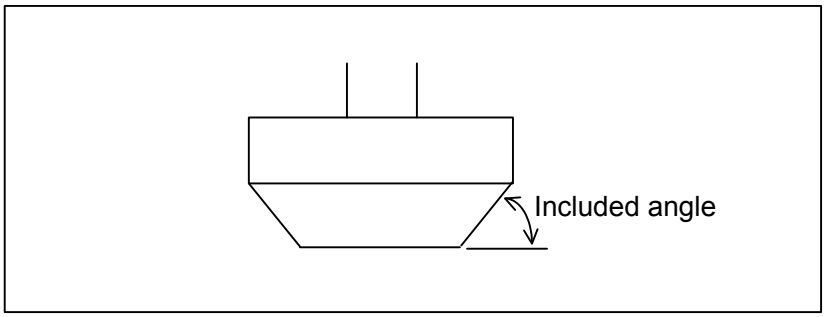
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the length of cut applied when a tapping tool is drawn in animated simulation.



When 0 is set, 50 mm for metric input or 1.9685 inch for inch input is assumed.

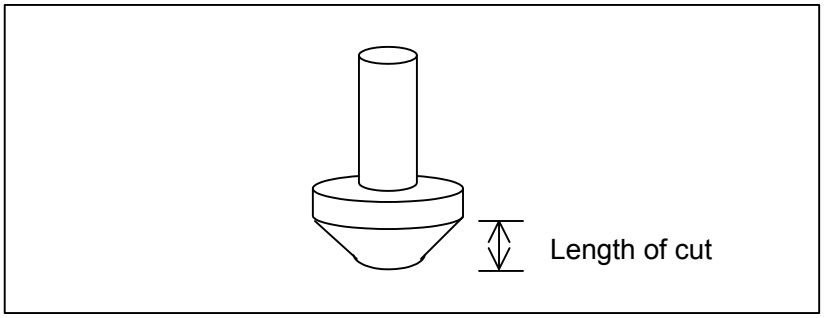
27375	Included angle applied when a chamfering tool is drawn in animated simulation

[Input type] Parameter input
[Data type] 2-word
[Unit of data] degree
[Valid data range] 0 to 90
This parameter sets the included angle applied when a a chamfering tool is drawn in animated simulation.



27376	Length of cut applied when a chamfering tool is drawn in animated simulation

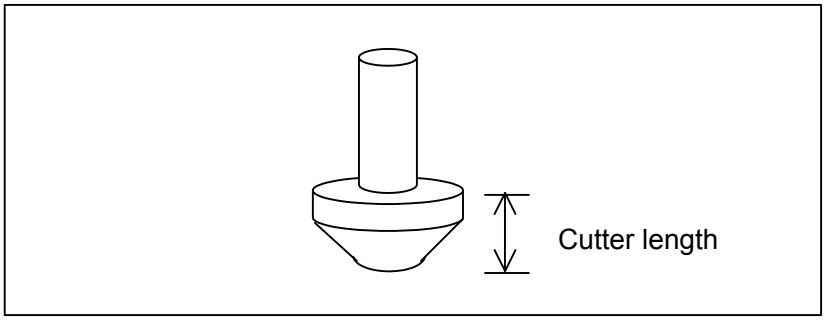
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the length of cut applied when a chamfering tool is drawn in animated simulation.



When 0 is set, 26 mm for metric input or 1.0236 inch for inch input is assumed.

27377	Cutter length applied when a chamfering tool is drawn in animated simulation

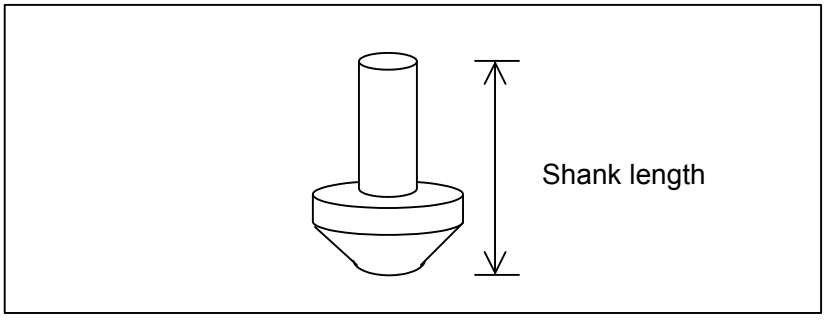
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the cutter length applied when a chamfering tool is drawn in animated simulation.



When 0 is set, 50 mm for metric input or 1.9685 inch for inch input is assumed.

27378	Shank length applied when a chamfering tool is drawn in animated simulation

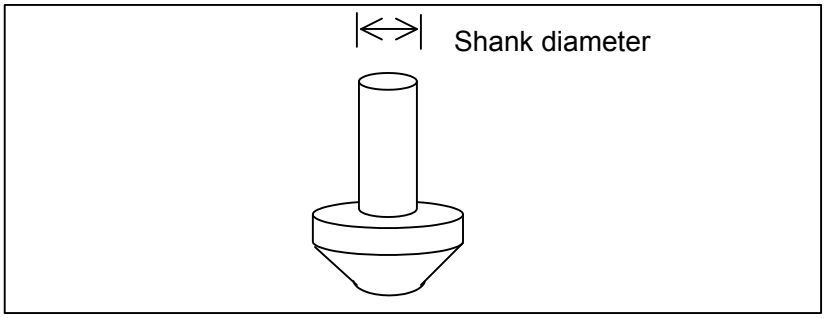
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the shank length applied when a chamfering tool is drawn in animated simulation.



When 0 is set, 130 mm for metric input or 5.1181 inch for inch input is assumed.

27379	Shank diameter applied when a chamfering tool is drawn in animated simulation

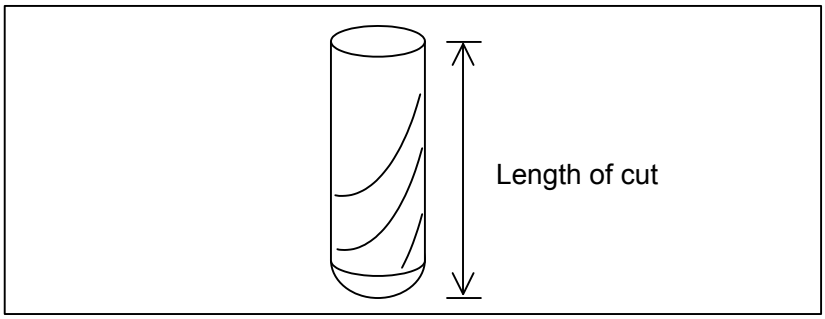
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the shank diameter applied when a chamfering tool is drawn in animated simulation.



When 0 is set, 32 mm for metric input or 1.2598 inch for inch input is assumed.

27380	Length of cut applied when a ball end mill is drawn in animated simulation

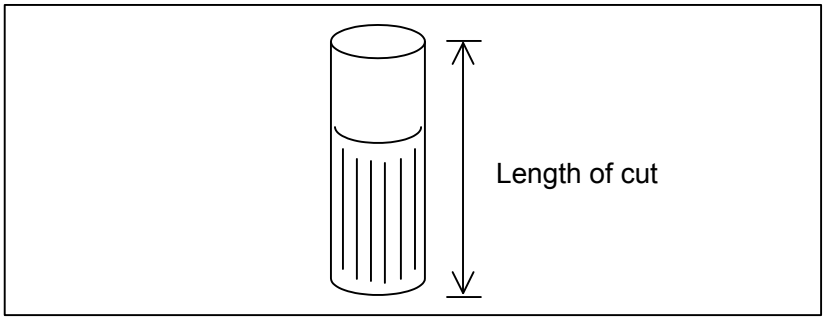
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the length of cut applied when a ball end mill is drawn in animated simulation.



When 0 is set, 50 mm for metric input or 1.9685 inch for inch input is assumed.

27381	Length of cut applied when a reamer is drawn in animated simulation

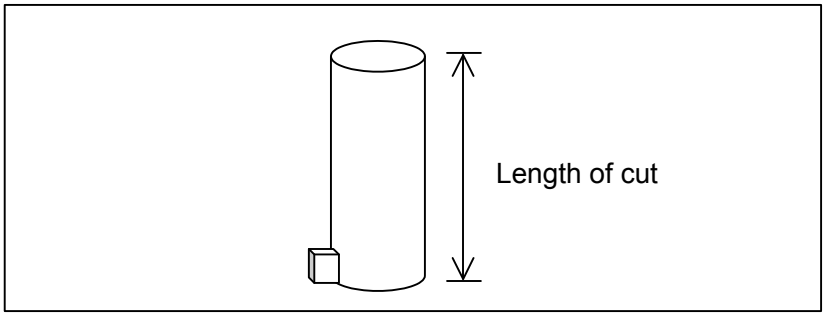
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the length of cut applied when a reamer is drawn in animated simulation.



When 0 is set, 50 mm for metric input or 1.9685 inch for inch input is assumed.

27382	Length of cut applied when a boring tool is drawn in animated simulation

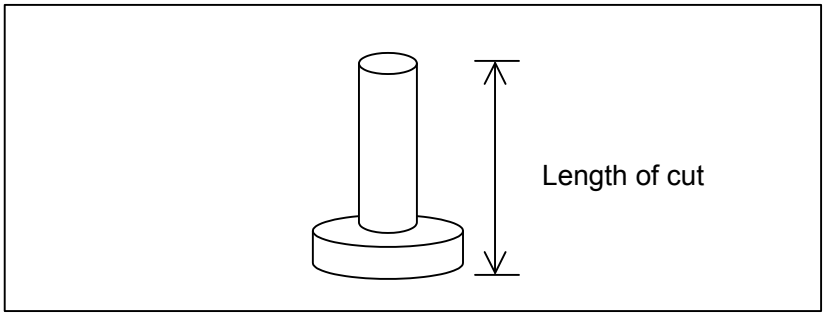
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the length of cut applied when a boring tool is drawn in animated simulation.



When 0 is set, 50 mm for metric input or 1.9685 inch for inch input is assumed.

27383	Length of cut applied when a face milling cutter is drawn in animated simulation

[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the length of cut applied when a face milling cutter is drawn in animated simulation.

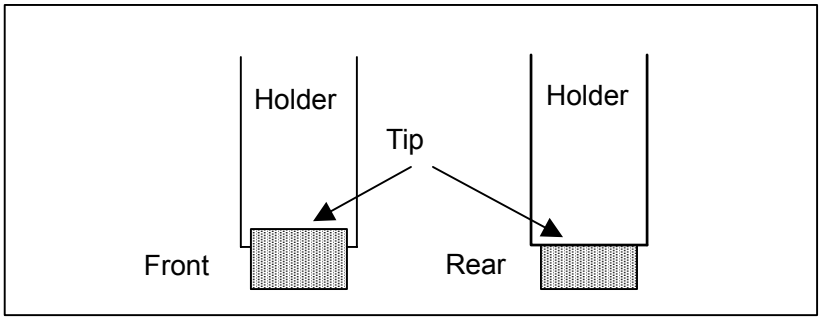


When 0 is set, 63 mm for metric input or 2.4803 inch for inch input is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
27384								VRP

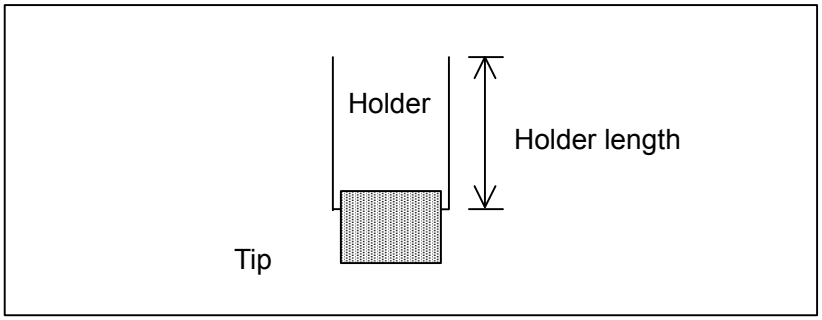
[Input type] Parameter input
[Data type] Bit

#0 VRP When a multifunction tool is drawn in animated simulation, the tip is:
0: Positioned on the front.
1: Positioned on the rear.



27385	Holder length applied when a multifunction tool is drawn in animated simulation

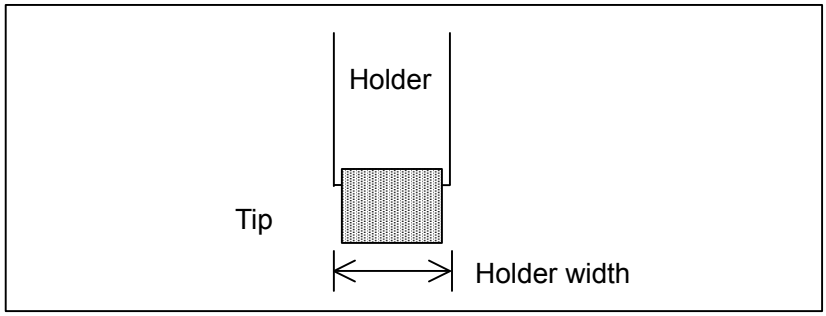
[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the holder length applied when a multifunction tool is drawn in animated simulation.



When 0 is set, 50 mm for metric input or 1.9685 inch for inch input is assumed.

27386	Holder width applied when a multifunction tool is drawn in animated simulation

[Input type] Parameter input
[Data type] 2-word
[Unit of data] 0.001mm (metric input), 0.0001inch (inch input)
[Valid data range] 0 or larger
This parameter sets the holder width applied when a multifunction tool is drawn in animated simulation.



When 0 is set, 14 mm for metric input or 0.5512 inch for inch input is assumed.

A.2 DATA TYPE

Parameters are classified by data type as follows:

Data type	Valid data range	Remarks
Bit	0 or 1	
Bit machine group		
Bit path		
Bit axis		
Bit spindle		
Byte	-128 to 127 0 to 255	Some parameters handle these types of data as unsigned data.
Byte machine group		
Byte path		
Byte axis		
Byte spindle		
Word	-32768 to 32767 0 to 65535	Some parameters handle these types of data as unsigned data.
Word machine group		
Word path		
Word axis		
Word spindle		
2-word	0 to ±999999999	Some parameters handle these types of data as unsigned data.
2-word machine group		
2-word path		
2-word axis		
2-word spindle		
Real	See the standard parameter setting tables.	
Real machine group		
Real path		
Real axis		
Real spindle		

NOTE

1 Each of the parameters of the bit, bit machine group, bit path, bit axis, and bit spindle types consists of 8 bits for one data number (parameters with eight different meanings).
2 For machine group types, parameters corresponding to the maximum number of machine groups are present, so that independent data can be set for each machine group.
3 For path types, parameters corresponding to the maximum number of paths are present, so that independent data can be set for each path.
4 For axis types, parameters corresponding to the maximum number of control axes are present, so that independent data can be set for each control axis.
5 For spindle types, parameters corresponding to the maximum number of spindles are present, so that independent data can be set for each spindle axis.
6 The valid data range for each data type indicates a general range. The range varies according to the parameters. For the valid data range of a specific parameter, see the explanation of the parameter.

A.3 STANDARD PARAMETER SETTING TABLES

This section defines the standard minimum data units and valid data ranges of the CNC parameters of the real type, real machine group type, real path type, real axis type, and real spindle type. The data type and unit of data of each parameter conform to the specifications of each function.

NOTE
1 Values are rounded up or down to the nearest multiples of the minimum data unit.
2 A valid data range means data input limits, and may differ from values representing actual performance.
3 For information on the ranges of commands to the CNC, refer to Appendix D, "Range of Command Value."

(A) Length and angle parameters (type 1)

Unit of data	Increment system	Minimum data unit	Valid data range
mm deg.	IS-A	0.01	-999999.99 to +999999.99
	IS-B	0.001	-999999.999 to +999999.999
	IS-C	0.0001	-99999.9999 to +99999.9999
	IS-D	0.00001	-9999.99999 to +9999.99999
	IS-E	0.000001	-999.999999 to +999.999999
inch	IS-A	0.001	-99999.999 to +99999.999
	IS-B	0.0001	-99999.9999 to +99999.9999
	IS-C	0.00001	-9999.99999 to +9999.99999
	IS-D	0.000001	-999.999999 to +999.999999
	IS-E	0.0000001	-99.9999999 to +99.9999999

(B) Length and angle parameters (type 2)

Unit of data	Increment system	Minimum data unit	Valid data range
mm deg.	IS-A	0.01	0.00 to +999999.99
	IS-B	0.001	0.000 to +999999.999
	IS-C	0.0001	0.0000 to +99999.9999
	IS-D	0.00001	0.00000 to +9999.99999
	IS-E	0.000001	0.000000 to +999.999999
inch	IS-A	0.001	0.000 to +99999.999
	IS-B	0.0001	0.0000 to +99999.9999
	IS-C	0.00001	0.00000 to +9999.99999
	IS-D	0.000001	0.000000 to +999.999999
	IS-E	0.0000001	0.0000000 to +99.9999999

(C) Velocity and angular velocity parameters

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min degree/min	IS-A	0.01	0.00 to +999000.00
	IS-B	0.001	0.000 to +999000.000
	IS-C	0.0001	0.0000 to +99999.9999
	IS-D	0.00001	0.00000 to +9999.99999
	IS-E	0.000001	0.000000 to +999.999999
inch/min	IS-A	0.001	0.000 to +96000.000
	IS-B	0.0001	0.0000 to +9600.0000
	IS-C	0.00001	0.00000 to +4000.00000
	IS-D	0.000001	0.000000 to +400.000000
	IS-E	0.0000001	0.0000000 to +40.0000000

(D) Acceleration and angular acceleration parameters

Unit of data	Increment system	Minimum data unit	Valid data range
mm/sec^2 deg./sec^2	IS-A	0.01	0.00 to +999999.99
	IS-B	0.001	0.000 to +999999.999
	IS-C	0.0001	0.0000 to +99999.9999
	IS-D	0.00001	0.00000 to +9999.99999
	IS-E	0.000001	0.000000 to +999.999999
inch/sec^2	IS-A	0.001	0.000 to +99999.999
	IS-B	0.0001	0.0000 to +99999.9999
	IS-C	0.00001	0.00000 to +9999.99999
	IS-D	0.000001	0.000000 to +999.999999
	IS-E	0.0000001	0.0000000 to +99.9999999

B PROGRAM CODE LIST

Character name	ISO code		EIA code		Custom macro		Usable as file name
	Character	Code (hexadecimal)	Character	Code (hexadecimal)	Without custom macro	With custom macro	
Number 0	0	30	0	20			*
Number 1	1	B1	1	01			*
Number 2	2	B2	2	02			*
Number 3	3	33	3	13			*
Number 4	4	B4	4	04			*
Number 5	5	35	5	15			*
Number 6	6	36	6	16			*
Number 7	7	B7	7	07			*
Number 8	8	B8	8	08			*
Number 9	9	39	9	19			*
Address A	A	41	a	61			*
Address B	B	42	b	62			*
Address C	C	C3	c	73			*
Address D	D	44	d	64			*
Address E	E	C5	e	75			*
Address F	F	C6	f	76			*
Address G	G	47	g	67			*
Address H	H	48	h	68			*
Address I	I	C9	i	79			*
Address J	J	CA	j	51			*
Address K	K	4B	k	52			*
Address L	L	CC	l	43			*
Address M	M	4D	m	54			*
Address N	N	4E	n	45			*
Address O	O	CF	o	46			*
Address P	P	50	p	57			*
Address Q	Q	D1	q	58			*
Address R	R	D2	r	49			*
Address S	S	53	s	32			*
Address T	T	D4	t	23			*
Address U	U	55	u	34			*
Address V	V	56	v	25			*
Address W	W	D7	w	26			*
Address X	X	D8	x	37			*
Address Y	Y	59	y	38			*
Address Z	Z	5A	z	29			*
Delete	DEL	FF	Del	7F	×	×	
Back space	BS	88	BS	2A	×	×	
Tabulator	HT	09	Tab	2E	×	×	
End of block	LF or NL	0A	CR or EOB	80			
Carriage return	CR	8D			×	×	
Space	SP	A0	SP	10	□	□	
Absolute rewind stop	%	A5	ER	0B			
Control out (start of comment)	(	28	(2-4-5)	1A			
Control in (end of comment)	)	A9	(2-4-7)	4A			

Character name	ISO code		EIA code		Custom macro		Usable as file name
	Character	Code (hexadecimal)	Character	Code (hexadecimal)	Without custom macro	With custom macro	
Plus sign	+	2B	+	70			*
Minus sign	-	2D	-	40			*
Colon (address O)	:	3A					
Optional block skip	/	AF	/	31			
Period (decimal point)	.	2E	.	6B			*
Sharp	#	A3	Parameter (No.6012)				
Dollar sign	$	24			□	□	
Ampersand	&	A6	&	0E			
Apostrophe	‘	27			□	□	
Asterisk	*	AA	Parameter (No.6010)				
Comma	,	AC	,	3B			
Semicolon	;	BB			□	□	
Left angle bracket	<	3C					
Equal sign	=	BD	Parameter (No.6011)				
Right angle bracket	>	BE					
Question mark	?	3F			△	○	
Commercial at mark	@	C0			△	△	
Quotation mark	“	22			□	□	
Left square bracket	[	DB	Parameter (No.6013)		△		
Right square bracket	]	DD	Parameter (No.6014)		△		
Underscore	_	6F	Parameter (No.6018)		△	△	*
Lowercase letter a	a	E1			△	△	*
Lowercase letter b	b	E2			△	△	*
Lowercase letter c	c	63			△	△	*
Lowercase letter d	d	E4			△	△	*
Lowercase letter e	e	65			△	△	*
Lowercase letter f	f	66			△	△	*
Lowercase letter g	g	E7			△	△	*
Lowercase letter h	h	E8			△	△	*
Lowercase letter i	i	69			△	△	*
Lowercase letter j	j	6A			△	△	*
Lowercase letter k	k	EB			△	△	*
Lowercase letter l	l	6C			△	△	*
Lowercase letter m	m	ED			△	△	*
Lowercase letter n	n	EE			△	△	*
Lowercase letter o	o	6F			△	△	*
Lowercase letter p	p	F0			△	△	*
Lowercase letter q	q	71			△	△	*
Lowercase letter r	r	72			△	△	*
Lowercase letter s	s	F3			△	△	*
Lowercase letter t	t	74			△	△	*
Lowercase letter u	u	F5			△	△	*
Lowercase letter v	v	F6			△	△	*
Lowercase letter w	w	77			△	△	*
Lowercase letter x	x	78			△	△	*

Character name	ISO code		EIA code		Custom macro		Usable as file name
	Character	Code (hexadecimal)	Character	Code (hexadecimal)	Without custom macro	With custom macro	
Lowercase letter y	y	F9			△	△	*
Lowercase letter z	z	FA			△	△	*

NOTE

1 The symbols used in the "Custom macro" column have the following meanings.
(Space) : The character will be registered in memory and has a specific meaning. It is used incorrectly in a statement other than a comment, an alarm occurs.
× : The character will not be registered in memory and will be ignored.
△ : The character will be registered in memory, but will be ignored during program execution. The character that has "*" in its "Usable as file name" field and is used as a file name is not ignored. Lowercase alphabetic characters in an NC program follow the setting of bit 0 (ESL) of parameter No. 3459.
○ : The character will be registered in memory. If it is used in a statement other than a comment, an alarm occurs.
□ : If it is used in a statement other than a comment, the character will not be registered in memory. If it is used in a comment, it will be registered in memory.

2 The symbol used in the column "Usable as file name" has the following meaning:
* : Capable of being coded between "<" and ">" as a file name.

3 Codes not in this table are ignored if their parity is correct.

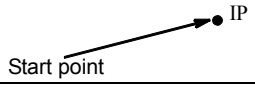
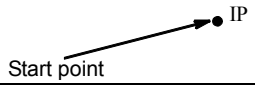
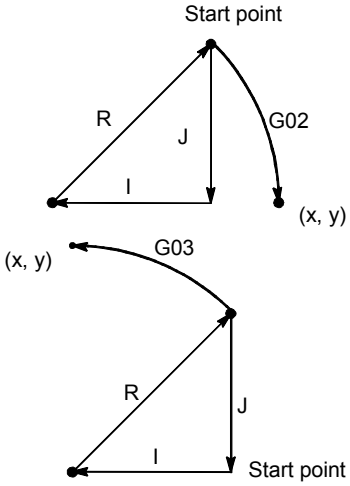
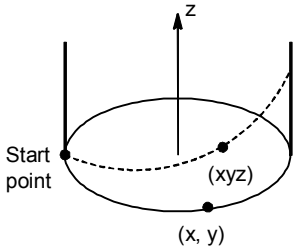
4 Codes with incorrect parity cause the TH alarm. But they are ignored without generating the TH alarm when they are in the comment section.

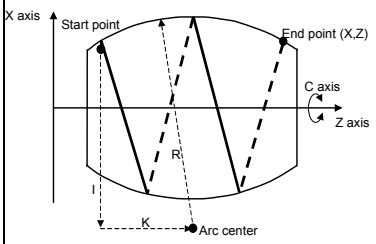
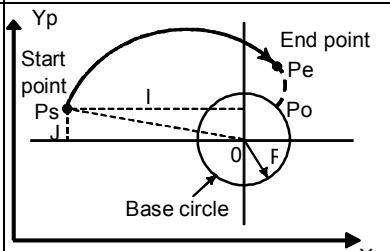
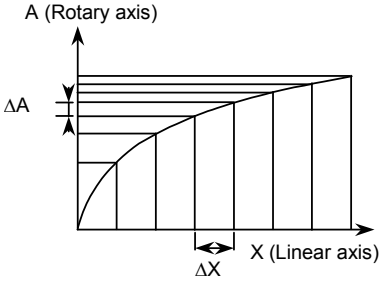
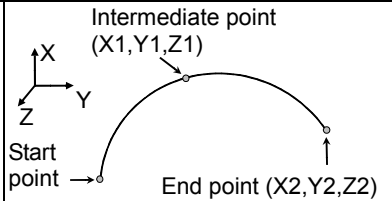
C LIST OF FUNCTIONS AND PROGRAM FORMAT

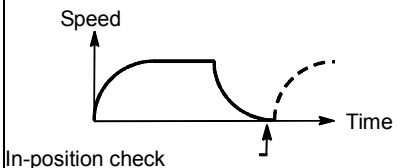
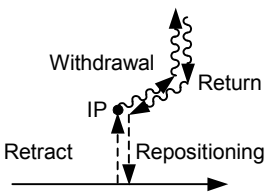
With some functions, the format used for specification on the machining center system differs from the format used for specification on the lathe system. Moreover, some functions are used for only one of the control types for the machining center system and lathe system.
Some functions cannot be added as options depending on the model.
For details of command formats, see the relevant sections or subsections.

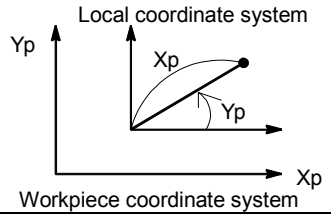
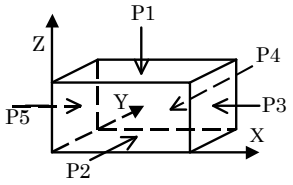
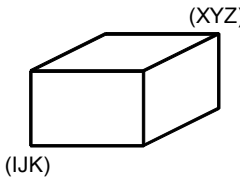
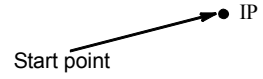
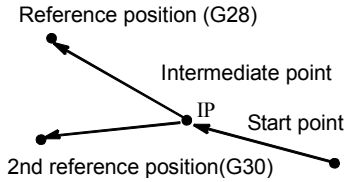
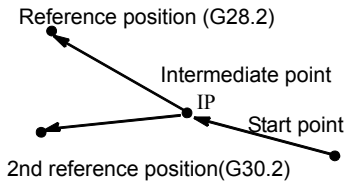
In the list, the following symbols are used:

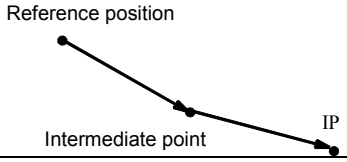
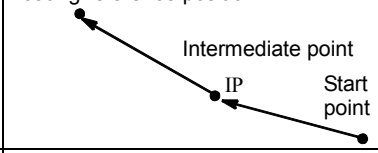
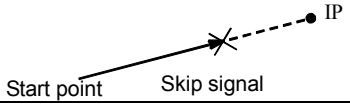
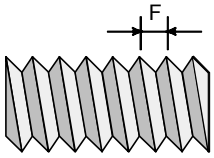
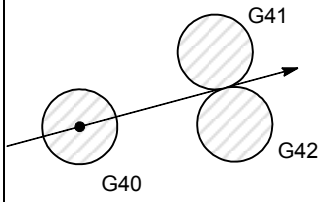
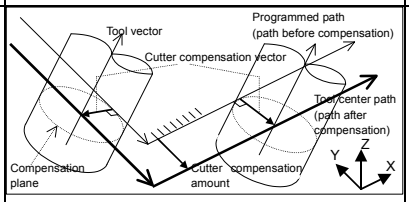
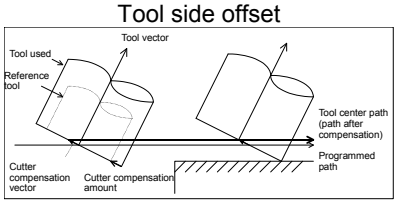
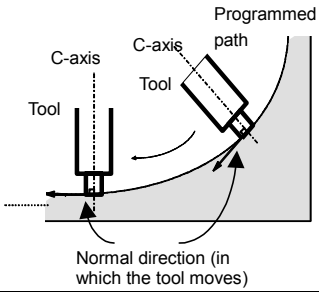
- For machining center
  x : 1st basic axis (X),
  y : 2nd basic axis (Y),
  z : 3rd basic axis (Z)
- For lathe
  x : 1st basic axis (X),
  z : 2nd basic axis (Z),
  coded using G code system A
- IP_ : presents a combination of arbitrary axis addresses using X,Y,Z,A,B and C (such as X_ Y_ Z_).

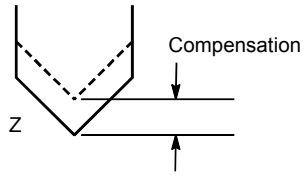
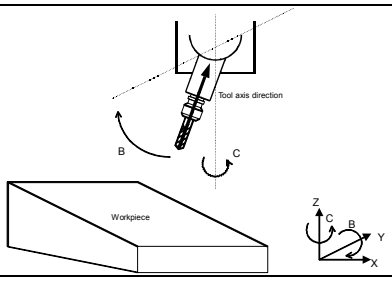
Functions	Illustration	Program format
Positioning (G00)		G00 IP_ ;
Linear interpolation (G01)		G01 IP_ F_;
Circular interpolation (G02, G03)		**• For machining center** G17 {G02 / G03} X_ Y_ {R_ / I_ J_} F_ ; G18 {G02 / G03} X_ Z_ {R_ / I_ K_} F_ ; G19 {G02 / G03} Y_ Z_ {R_ / J_ K_} F_ ; **• For lathe** {G02 / G03} X_ Z_ {R_ / I_ K_} F_ ;
Helical interpolation (G02, G03)	 In case of G03 on X-Y plane	G17 {G02 / G03} X_ Y_ {R_ / I_ J_} α_ F_ ; G18 {G02 / G03} X_ Z_ {R_ / I_ K_} α_ F_ ; G19 {G02 / G03} Y_ Z_ {R_ / J_ K_} α_ F_ ; α: Arbitrary address except the circular interpolation axis

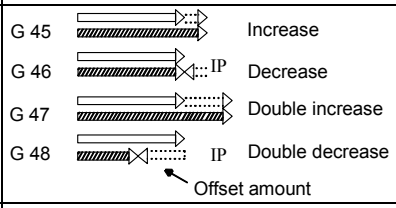
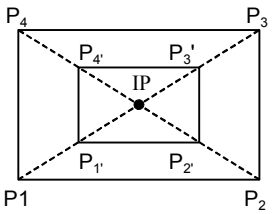
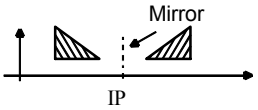
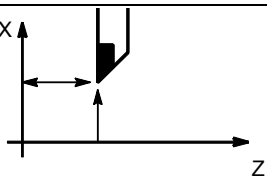
Functions	Illustration	Program format
Circular thread cutting B (G02.1, G03.1)	 In the case of the ZpXp plane, the major axis being the Z-axis, the minor axis being the X-axis, the arbitrary axis being the C-axis, and clockwise direction	G17 {G02.1 / G03.1} X_ Y_ α_ β_ {I_ J_ / R_} F_ ; G18 {G02.1 / G03.1} Z_ X_ α_ β_ {K_ I_ / R_} F_ ; G19 {G02.1 / G03.1} Y_ Z_ α_ β_ {J_ K_ / R_} F_ ; X,Y,Z : Coordinates of the end point for circular interpolation α, β : Coordinates of the end point for linear interpolation I,J,K : Distance from the start point to the center of an arc R : Arc radius F : Feedrate in the major axis direction α and β are arbitrary axes other than the circular interpolation axis. Up to two such axes can be specified.
Involute interpolation (G02.2, G03.2)	 (In case of X-Y plane)	G17 {G02.2 / G03.2} Xp_ Yp_ I_ J_ R_ F_ ; G18 {G02.2 / G03.2} Zp_Xp_ K_ I_ R_ F_ ; G19 {G02.2 / G03.2} Yp_ Zp_ J_ K_ R_ F_ ;
Exponential interpolation (G02.3, G03.3)	 Relationship between X-axis and A-axis	Positive rotation G02.3 X_ Y_ Z_ I_ J_ K_ R_ F_ Q_; Negative rotation G03.3 X_ Y_ Z_ I_ J_ K_ R_ F_ Q_;
3-dimensional circular interpolation (G02.4, G03.4)		G02.4 X_{X1} Y_{Y1} Z_{Z1} $\alpha_{\alpha1}$ $\beta_{\beta1}$; First block (mid-point of the arc) X_{X2} Y_{Y2} Z_{Z2} $\alpha_{\alpha2}$ $\beta_{\beta2}$; Second block (end point of the arc) α, β : Arbitrary axes other than the 3-dimensional circular interpolation axis (up to two axes) G03.4 can also be specified instead of G02.4.
Dwell (G04)		G04 {X_ / P_} ;
High-speed cycle machining (G05)		G05 P10xxx Lxxx ; P10xxx : Start number of the machining cycle to call (P10001 to P10999) Lxxx : Repeat count of the machining cycle (L1 to L999 / assumed to be L1 if omitted)

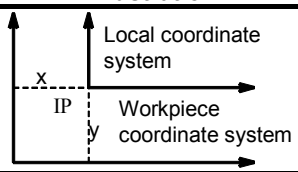
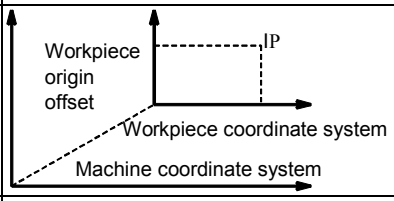
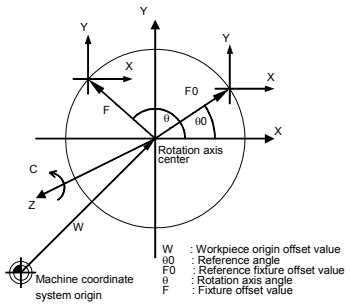
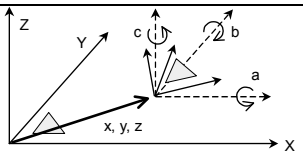
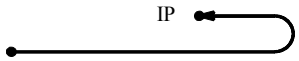
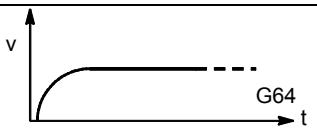
Functions	Illustration	Program format
AI contour control (G05)		G05 P10000 ; AI contour control start G05 P0 ; AI contour control end
AI contour control (G05.1)		G05.1 Q1 ; AI contour control mode on G05.1 Q0 ; AI contour control mode off
Nano smoothing (G05.1)		G05.1 Q3 IP0 ; Nano smoothing mode on G05.1 Q0 ; Nano smoothing mode off
Smooth interpolation (G05.1)		G05.1 Q2 ; Smooth interpolation mode on G05.1 Q0 ; Smooth interpolation mode off
NURBS interpolation (G06.2)		G06.2[P_] K_ IP_ [R_] [F_]; NURBS interpolation mode on P : Rank of NURBS curve IP : Control point R : Weight K : Knot F : Feedrate
Hypothetical axis interpolation (G07)		G07 IP0 ; Hypothetical axis setting G07 IP1 ; Hypothetical axis cancel
Cylindrical interpolation (G07.1)		G07 IP_ r_; Cylindrical interpolation mode r : Cylinder radius G07 IP 0; Cylindrical interpolation mode cancel
AI contour control (Advanced preview control) (G08)		G08 P1 ; AI contour control mode on G08 P0 ; AI contour control mode off
Exact stop (G09)		G09 { G01 / G02 / G03 } IP_ ;
Programmable data input (G10)		• **For machining center** Tool compensation memory A G10 L01 P_ R_ ; Tool compensation memory B G10 L10 P_ R_ ; (Geometry offset amount) G10 L11 P_ R_ ; (Wear offset amount) Tool compensation memory C G10 L10 P_ R_ ; (Geometry offset amount/H) G10 L11 P_ R_ ; (Wear offset amount/H) G10 L12 P_ R_ ; (Geometry offset amount/D) G10 L13 P_ R_ ; (Wear offset amount/D) • **For lathe** Geometry offset amount G10 P_ X_ Z_ R_ Q_ ; P = 10000 + Geometry offset number Wear offset amount G10 P_ X_ Z_ C_ Q_ ; P = Wear offset number
Tool retract and recover (G10.6)		G10.6 IP_ ; Specify the amount of retraction G10.6 (as a single block containing no other commands) ; Cancel the amount of retraction

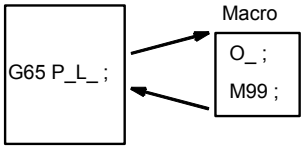
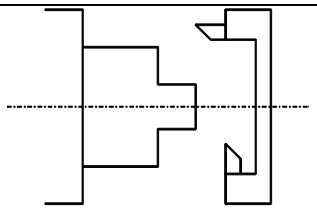
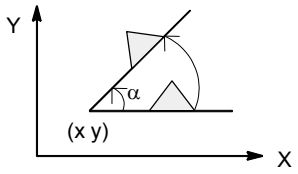
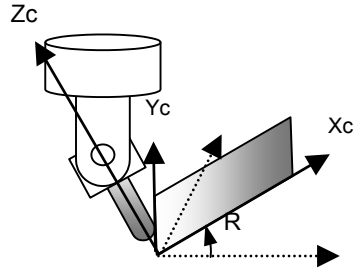
Functions	Illustration	Program format
Programmable internal data change (G10.8)		Tolerance change in smooth TCP mode G10.8 L1 {α_ β_ / P_} ; α : Compensation tolerance for the first rotation axis β : Compensation tolerance for the second rotation axis P0 : 0 is used as the compensation tolerance for first or second rotation axis P1 : Parameter No.10486 or 10487 is used as the compensation tolerance for first or second rotation axis
Polar coordinate interpolation (G12.1, G13.1)		G12.1 ; Polar coordinate interpolation mode on G13.1 ; Polar coordinate interpolation cancel
Polar coordinate command (G15, G16)		G17 G16 Xp_ Yp_ . . . ; G18 G16 Zp_ Xp_ . . . ; G19 G16 Yp_ Zp_ . . . ; G15 ; Cancel
Plane selection (G17, G18, G19)		G17 ; Xp Yp-plane selection G18 ; Zp Xp-plane selection G19 ; Yp Zp-plane selection
Plane conversion (G17.1)		G17.1 P_ ; P : Plane conversion specification
Inch/metric conversion (G20, G21)		Inch input G20 ; Metric input G21 ;
Stored stroke check (G22, 23)		G22 X_ Y_ Z_ I_ J_ K_ ; G23 ; Cancel
Reference position return check (G27)		G27 IP_ ;
Reference position return (G28) 2nd Reference position return (G30)		G28 IP_ ; G30 IP_ ;
In-position check disable reference position return (G28.2) In-position check disable 2nd reference position return (G30.2)		G28.2 IP_; G30.2 IP_;

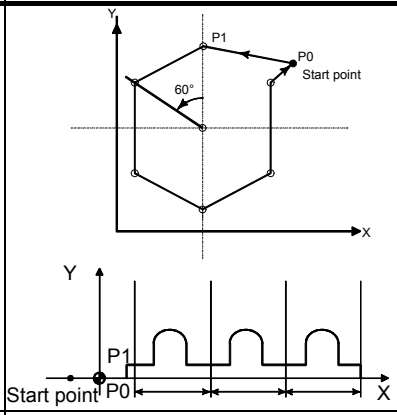
Functions	Illustration	Program format
Movement from reference position (G29)		G29 IP_ ;
Floating reference position return (G30.1)		G30.1 IP_ ;
Skip function (G31)		G31 IP_ F_ ;
Threading (G33)		• **For machining center** G33 IP_ F_ ; F : Lead
Threading (G32)		• **For lathe** Equal lead threading G32 P_ F_ ;
Tool radius/tool nose radius compensation, 3-dimensional cutter compensation (G38, G39, G40 to G42)		• **For machining center** {G17, G18, G19} {G41, G42} D_ ; D : Tool compensation number G40 : Cancel
Tool radius/tool nose radius compensation (G40 to G42) (G38, G39)		• **For lathe only** {G41, G42} IP_ ; G40 : Cancel
3-dimensional cutter compensation (G41.2, G42.2)	 Tool side offset	{G41.2, G42.2} IP_ D_ ; Tool side offset G40 IP_ ; Cancellation of cutter compensation
(G41.3)	 Leading edge offset	G41.3 D_ ; Leading edge offset G40 ; Canceling the leading edge offset
Normal direction control (G40.1, G41.1, G42.1)		G41.1 ; Normal direction control on : right G42.1 ; Normal direction control on : left G40.1 ; Normal direction control cancel

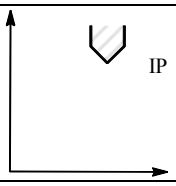
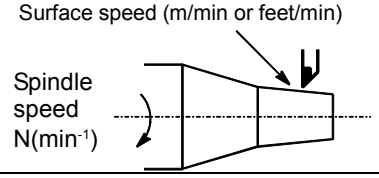
Functions	Illustration	Program format
Tool length compensation (G43, G44, G49)		{G43 / G44} Z_ H_ ; {G43 / G44} H_ ; H : Tool compensation number G49 : Cancel
Tool length compensation in tool axis direction (G43.1)		G43.1 H_ ; Tool length compensation in tool axis direction H : Compensation number G49 ; Compensation cancel
Tool center point control (TYPE1) (G43.4)		G43.4 IP α β H ; Tool center point control (TYPE1) start IP α β ; IP : In the case of an absolute command, the coordinate value of the end point of the tool tip movement In the case of an incremental command, the amount of the tool tip movement α, β : In the case of an absolute command, the coordinate value of the end point of the rotation axes In the case of an incremental command, the amount of the rotation axis movement H : Tool compensation number
Tool center point control (TYPE2) (G43.5)		G43.5 IP_ H_ Q_ ; Tool center point control (TYPE2) start IP_ I_ J_ K_ ; IP : In the case of an absolute command, the coordinate value of the end point of the tool tip movement In the case of an incremental command, the amount of the tool tip movement I,J,K : Tool axis direction at the block end point as seen from the programming coordinate system H : Tool compensation number Q : Inclination angle of the tool (in degrees)
Tool offset (G43.7)		• **For lathe** G43.7 D_ ; D : Tool compensation number • **For machining center** G43.7 H_ ; H : Tool compensation number

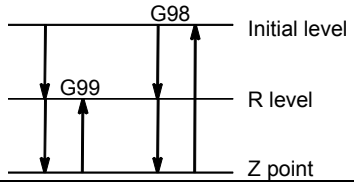
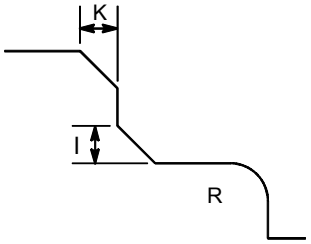
Functions	Illustration	Program format
Tool offset conversion (G44.1)		• **For lathe** G44.1 D_ P_ ; D : Tool compensation number P : Direction of imaginary tool nose (P_ can be omitted.) • **For machining center** G43.7 H_ P_ ; H : Tool compensation number P : Direction of imaginary tool nose (P_ can be omitted.)
Tool offset (G45 to G48)		• **For machining center** {G45, G46, G47, G48} IP_ D_ ; D : Tool offset number
Scaling (G50, G51)		• **For machining center** G51 X_ Y_ Z_ {P_ / I_ J_ K_} ; P, I, J, K : Scaling magnification X, Y, Z : Control position of scaling G50 : Cancel • **For lathe only** Enabled when G code system B/C is used
Programmable mirror image (G50.1, G51.1)		G51.1 IP_ ; G50.1 ; . . . Cancel
Synchronous, composite, and superimposed control by program command (G50.4, G51.4, G50.5, G51.5, G50.6, G51.6)		G51.4 P_Q_(L_) ; Start synchronous control (L_ can be omitted.) G50.4 Q_ ; Cancel synchronous control. P : Number to identify synchronous master axis Q : Number to identify synchronous slave axis L : Parking start command G51.5 P_Q_ ; Start composite control G50.5 P_Q_ ; Cancel composite control P : Number to identify composite axis 1 Q : Number to identify composite axis 2 G51.6 P_Q_ ; Start superimposed control G50.6 Q_ ; Cancel superimposed control P : Number to identify superimposed master axis Q : Number to identify superimposed slave axis
Coordinate system setting Maximum spindle speed clamp (G50)		• **For lathe only** G50 IP_ ; (Coordinate system setting) G50 S_ ; (Maximum spindle speed clamp)

Functions	Illustration	Program format
Local coordinate system setting (G52)		G52 IP_ ;
Command in machine coordinate system (G53)		G53 IP_ ;
Tool axis direction control (G53.1)		G53.1 ; Tool axis direction control
Selection of workpiece coordinate system (G54 to G59)		{G54 : G59} IP_ ;
Rotary table dynamic fixture offset (G54.2)		• **For machining center** G54.2 P_ ; Fixture offset P : Reference fixture offset value number G54.2 P0 ; Offset cancel
Workpiece setting error compensation (G54.4)	 x : X-axis direction error y : Y-axis direction error z : Z-axis direction error a : Rotation direction error (about the X-axis) b : Rotation direction error (about the Y-axis) c : Rotation direction error (about the Z-axis)	G54.4 P_ ; Workpiece setting error compensation P : Workpiece setting error number G54.4 P0 ; Workpiece setting error compensation cancel
Single direction positioning (G60)		G60 IP_ ;
Cutting mode (G64) Exact stop mode (G61) Tapping mode (G63)		G64_ ; Cutting mode G61_ ; Exact stop mode G63_ ; Tapping mode
Automatic corner override (G62)		G62_ ; Automatic corner override

Functions	Illustration	Program format
Custom macro (G65, G66, G66.1, G67)		• **One-shot call** G65 P_ L_ <Argument assignment> ; P : Program number L : Number of repetition • **Modal call** G66 P_ L_ <Argument assignment> ; Call after the move command G66.1 P_ L_ <Argument assignment> ; Each block call G67 ; Cancel
Mirror image for double turret (G68, G69)		• **For lathe only** G68 : Mirror image for double turret G69 : Mirror image cancel
Coordinate system rotation, 3-dimensional coordinate conversion (G68, G69) (G68.1, G69.1)	 In case of X-Y plane	• **For machining center** G68 {G17 X_ Y_ / G18 Z_ X_ / G19 Y_ Z_} R α ; G69 ; Cancel • **For lathe** G68.1 {G17 X_ Y_ / G18 Z_ X_ / G19 Y_ Z_} R α ; G69.1 ; Cancel
Tilted working plane indexing with guidance (G68.2, G68.4)		G68.2/G68.4 P_ X_ Y_ Z_ I_ J_ K_ ; G68.2 : Tilted working plane indexing G68.4 : Tilted working plane indexing (incremental multiplexed command) G69 ; Tilted working plane command cancel Without P : Euler's angle P1 : Roll, pitch, yaw P2 : Three points P3 : Two vectors P4 : Projection angle X, Y, Z : Feature coordinate system origin I, J, K : Euler angles for determining the orientation of the feature coordinate system
Tilted working plane indexing by tool axis direction (G68.3)	 Feature coordinate system Xc-Yc-Zc	G68.3 X_ Y_ Z_ R_ ; G69 ; Cancel X,Y, Z : Feature coordinate system origin R : Feature coordinate system rotation angle about the Z-axis

Functions	Illustration	Program format
Figure copy (G72.1, G72.2)		• **Rotational copy** {(G17) (G18) (G19)} G72.1 P_ L_ {X_ Y_ Z_ X_ Y_ Z_} R_ ; • **Linear copy** {(G17) (G18) (G19)} G72.2 P_ L_ {I_ J_ K_ I_ J_ K_} ;
Canned cycle for drilling (G73, G74, G80 to G89)		• **For machining center** G80 ; Cancel G73 G74 G76 G81 : G89 } X_Y_Z_P_Q_R_F_K_ ;
Canned cycle (G71 to G76) (G90, G92, G94)		• **For lathe only** N_ G70P_ Q_ ; G71U_ R_ ; G71P_ Q_ U_ W_ F_ S_ T_ ; G72W_ R_ ; G72P_ Q_ U_ W_ F_ S_ T_ ; G73U_ W_ R_ ; G73P_ Q_ U_ W_ F_ S_ T_ ; G74R_ ; G74X(u)_ Z(w)_ P_ Q_ R_ F_ ; G75R_ ; G75X(u)_ Z(w)_ P_ Q_ R_ F_ ; G76P_ Q_ R_ ; G76X(u)_ Z(w)_ P_ Q_ R_ F_ ; {G90 G92} X_ Z_ I_ F_ ; G94X_ Z_ K_ F_ ;
Canned grinding cycle (for grinding machine) (G71 to G75, G77 to G79)		• **For machining center** G75 I_ J_ K_ α_ R_ F_ P_ L_ ; G77 I_ J_ K_ α_ R_ F_ P_ L_ ; G78 I_ (J_) K_ α_ F_ P_ L_ ; G79 I_ J_ K_ α_ R_ F_ P_ L_ ; α : Arbitrary axis address of the grinding axis • **For lathe** G71 A_ B_ W_ U_ I_ K_ H_ ; G72 P_ A_ B_ W_ U_ I_ K_ H_ ; G73 A_ (B_) W_ U_ K_ H_ ; G74 P_ A_ (B_) W_ U_ K_ H_ ;

Functions	Illustration	Program format
Absolute/incremental programming (G90/G91)		• **For machining center** G90_ ; Absolute programming G91_ ; Incremental programming : G90_ G91_ ; Programming in both modes • **For lathe** X_ Z_ C_ : Absolute programming U_ W_ H_ : Incremental programming Distinguished by an address specified in combined use with a G function such as G00 and G01.
Maximum incremental command value check (G91.1)		G91.1 IP_ ; IP_ ; Maximum incremental value Set 0 to cancel maximum incremental value check.
Change of workpiece coordinate system (G92) Maximum spindle speed clamp (G92)		• **For machining center** G92 IP_ ; Change of workpiece coordinate system G92 S_ ; Constant surface speed control : Maximum spindle speed clamp
Workpiece coordinate system preset (G92.1)		• **For machining center** G92.1 IP 0 ;
Inverse time feed (G93)		G93 ; Inverse time setting mode
Feed per minute, Feed per revolution (G94, G95)	mm/min inch/min mm/rev inch/rev	• **For machining center** G94 F_ ; Feed per minute G95 F_ ; Feed per revolution
(G98, G99)		• **For lathe** G98 F_ ; Feed per minute G99 F_ ; Feed per revolution
Constant surface speed control (G96, G97)		G96 S_ ; Constant surface speed control on (surface speed specification) G97 S_ ; Constant surface speed control off (spindle speed specification)
Speed display function of a milling tool with servo motor (G96.1,G96.2,G96.3,G96.4)		G96.1 P_R_; The next block starts operating upon completion of spindle indexing (the SV speed control mode is off). G96.2 P_R_; The next block starts operating without waiting for spindle indexing to complete. G96.3 P_; The next block starts operating after confirming the completion of spindle indexing (the SV speed control mode is off). G96.4 P_; The SV speed control mode is on.

Functions	Illustration	Program format
Initial point return / R point return (G98, G99)		• **For machining center** G98_ ; G99_ ;
Optional chamfering/corner R		• **For machining center** ,C_ : Chamfering ,R_ : Corner R
Chamfering/corner R		• **For lathe only** X_ { C±K / R_ } P_; Z_ { C±K / R_ } P_;
In-feed control (for grinding machine) (G160, G161)		• **For machining center** G161 R_ ; Figure program (G01, G02, G03) G160 ;

D RANGE OF COMMAND VALUE

Linear axis

- In case of millimeter input, feed screw is millimeter

	Increment system				
	IS-A	**IS-B**	**IS-C**	**IS-D**	**IS-E**
Least input increment (mm)	0.01	0.001	0.0001	0.00001	0.000001
Least command increment (mm)	0.01	0.001	0.0001	0.00001	0.000001
Max. programmable dimension (mm)	±999,999.99	±999,999.999	±99,999.9999	±9,999.99999	±999.999999
Max. rapid traverse (mm/min)[*1]	999,000	999,000	100,000	10,000	1,000
Feedrate range (mm/min)[*1]	0.01 to 999,000	0.01 to 999,000	0.0001 to 100,000	0.00001 to 10,000	0.000001 to 1,000
Incremental feed (mm/step)	0.01 0.1 1.0 10.0	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1	0.00001 0.0001 0.001 0.01	0.000001 0.00001 0.0001 0.001
Tool compensation amount (mm)[*2]	0 to ±9,999.99	0 to ±9,999.999	0 to ±9,999.9999	0 to ±9,999.99999	0 to ±999.999999
Backlash compensation amount (pulses)[*3]	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)[*4]	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999	0 to 9,999.99999	0 to 999.999999

- In case of inch input, feed screw is millimeter

	Increment system				
	IS-A	**IS-B**	**IS-C**	**IS-D**	**IS-E**
Least input increment (inch)	0.001	0.0001	0.00001	0.000001	0.0000001
Least command increment (inch)	0.001	0.0001	0.00001	0.000001	0.0000001
Max. programmable dimension (inch)	±39,370.078	±39,370.0787	±3,937.00787	±393.700787	±39.3700787
Max. rapid traverse (mm/min)[*1]	999,000	999,000	100,000	10,000	1,000
Feedrate range (inch/min)[*1]	0.001 to 96,000	0.0001 to 9,600	0.00001 to 4,000	0.000001 to 400	0.0000001 to 40
Incremental feed (inch/step)	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1	0.00001 0.0001 0.001 0.01	0.000001 0.00001 0.0001 0.001	0.0000001 0.000001 0.00001 0.0001
Tool compensation amount (inch)[*2]	0 to ±999.999	0 to ±999.9999	0 to ±999.99999	0 to ±999.999999	0 to ±99.9999999
Backlash compensation amount (pulses)[*3]	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)[*4]	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999	0 to 9,999.99999	0 to 999.999999

- In case of inch input, feed screw is inch

	Increment system				
	IS-A	**IS-B**	**IS-C**	**IS-D**	**IS-E**
Least input increment (inch)	0.001	0.0001	0.00001	0.000001	0.0000001
Least command increment (inch)	0.001	0.0001	0.00001	0.000001	0.0000001

	Increment system				
	IS-A	**IS-B**	**IS-C**	**IS-D**	**IS-E**
Max. programmable dimension (inch)	±99,999.999	±99,999.9999	±9,999.99999	±999.999999	±99.9999999
Max. rapid traverse (inch/min)[*1]	96,000	9,600	4,000	400	40
Feedrate range (inch/min)[*1]	0.001 to 96,000	0.0001 to 9,600	0.00001 to 4,000	0.000001 to 400	0.0000001 to 40
Incremental feed (inch/step)	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1	0.00001 0.0001 0.001 0.01	0.000001 0.00001 0.0001 0.001	0.0000001 0.000001 0.00001 0.0001
Tool compensation amount (inch)[*4]	0 to ±999.999	0 to ±999.9999	0 to ±999.99999	0 to ±999.999999	0 to ±99.9999999
Backlash compensation amount (pulses)[*3]	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)[*4]	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999	0 to 9,999.99999	0 to 999.999999

- In case of millimeter input, feed screw is inch

	Increment system				
	IS-A	**IS-B**	**IS-C**	**IS-D**	**IS-E**
Least input increment (mm)	0.01	0.001	0.0001	0.00001	0.000001
Least command increment (mm)	0.01	0.001	0.0001	0.00001	0.000001
Max. programmable dimension (mm)	±999,999.99	±999,999.999	±99,999.9999	±9,999.99999	±999.999999
Max. rapid traverse (inch/min)[*1]	96,000	9,600	4,000	400	40
Feedrate range (mm/min)[*1]	0.01 to 999,000	0.001 to 999,000	0.0001 to 100,000	0.00001 to 10,000	0.000001 to 1,000
Incremental feed (mm/step)	0.01 0.1 1.0 10.0	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1	0.00001 0.0001 0.001 0.01	0.000001 0.00001 0.0001 0.001
Tool compensation amount (mm)[*2]	0 to ±9,999.99	0 to ±9,999.999	0 to ±9,999.9999	0 to ±9,999.99999	0 to ±999.999999
Backlash compensation amount (pulses)[*3]	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)[*4]	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999	0 to 9,999.99999	0 to 999.999999

- Rotary axis

	Increment system				
	IS-A	**IS-B**	**IS-C**	**IS-D**	**IS-E**
Least input increment (deg)	0.01	0.001	0.0001	0.00001	0.000001
Least command increment (deg)	0.01	0.001	0.0001	0.00001	0.000001
Max. programmable dimension (deg)	±999,999.99	±999,999.999	±99,999.9999	±9,999.99999	±999.999999
Max. rapid traverse (deg/min)[*1]	999,000	999,000	100,000	10,000	1,000
Feedrate range (deg/min)[*1]	0.01 to 999,000	0.001 to 999,000	0.0001 to 100,000	0.00001 to 10,000	0.000001 to 1,000
Incremental feed (deg/step)	0.01 0.1 1.0 10.0	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1	0.00001 0.0001 0.001 0.01	0.000001 0.00001 0.0001 0.001
Tool compensation amount (deg)[*2]	0 to ±9,999.99	0 to ±9,999.999	0 to ±9,999.9999	0 to ±9,999.99999	0 to ±999.999999

	Increment system				
	IS-A	**IS-B**	**IS-C**	**IS-D**	**IS-E**
Backlash compensation amount (pulses)*3	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)*4	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999	0 to 9,999.99999	0 to 999.999999

NOTE

*1 The feedrate range shown above are limitations depending on CNC interpolation capacity. As a whole system, limitations depending on servo system must also be considered.

*2 If the mode of input is switched between inch input and metric input, the maximum compensation value that can be set at inch input time is (maximum compensation value) × 1/25.4. If a value exceeding this value is specified at inch input time, the compensation value is not converted to a metric value correctly when the mode of input is switched to metric input.

*3 The unit is the detection unit.

*4 Depends on the increment system of the axis at in address X.

E NOMOGRAPHS

Appendix E, "NOMOGRAPHS", consists of the following sections:

E.1 INCORRECT THREADED LENGTH.....2167
E.2 SIMPLE CALCULATION OF INCORRECT THREAD LENGTH 2168
E.3 TOOL PATH AT CORNER 2170
E.4 RADIUS DIRECTION ERROR AT CIRCLE CUTTING 2173

E.1 INCORRECT THREADED LENGTH

The leads of a thread are generally incorrect in δ_1 and δ_2, as shown in Fig. E.1 (a), due to automatic acceleration and deceleration.

Thus distance allowances must be made to the extent of δ_1 and δ_2 in the program.

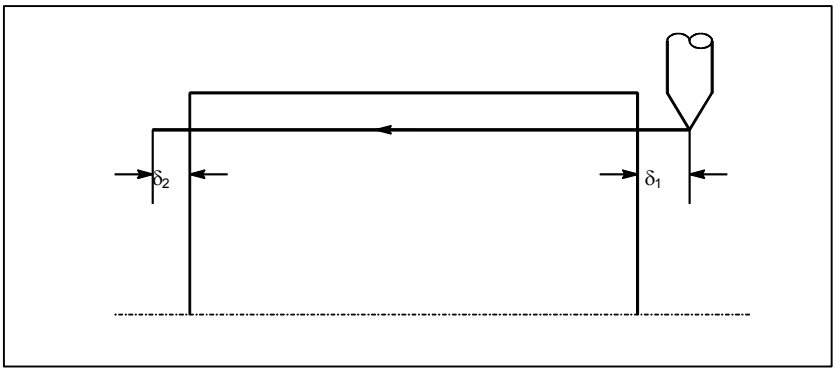


Fig. E.1 (a) Incorrect thread position

Explanation

- How to determine δ_2

$\delta_2 = T_1 V$ (mm). . . . (1)

$V = \frac{1}{60} RL$

T_1 : Time constant of servo system (sec)
V : Threading speed (mm/sec)
R : Spindle speed (min^{-1})
L : Thread feed (mm)
Time constant T_1 (sec) of the servo system: Usually 0.033 s.

- How to determine δ_1

$\delta_1 = \{t - T_1 + T_1 \exp(- \frac{t}{T_1})\} V$ . . . . . (2)

$a = \exp(-\frac{t}{T_1})$ . . . . . (3)

T_1 : Time constant of servo system (sec)
V : Threading speed (mm/sec)
Time constant T_1 (sec) of the servo system: Usually 0.033 s.

The lead at the beginning of thread cutting is shorter than the specified lead L, and the allowable lead error is ΔL. Then as follows.

$a = \frac{\Delta L}{L}$

When the value of "a" is determined, the time lapse until the thread accuracy is attained. The time "t" is substituted in (2) to determine δ_1: Constants V and T_1 are determined in the same way as for δ_2. Since the calculation of δ_1 is rather complex, a nomography is provided on the following pages.

- How to use nomograph

First specify the class and the lead of a thread. The thread accuracy, a, will be obtained at (1), and depending on the time constant of cutting feed acceleration/ deceleration, the δ_1 value when V = 10mm/s will be obtained at (2). Then, depending on the speed of thread cutting, δ_1 for speed other than 10mm/s can be obtained at (3).

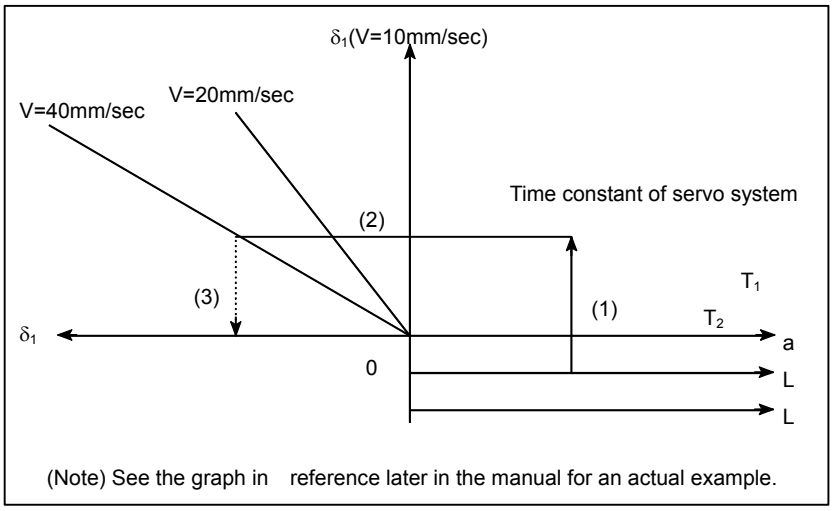


Fig. E.1 (b) Nomograph

NOTE
The equations for δ_1, and δ_2 are for when the acceleration/ deceleration time constant for cutting feed is 0.

E.2 SIMPLE CALCULATION OF INCORRECT THREAD LENGTH

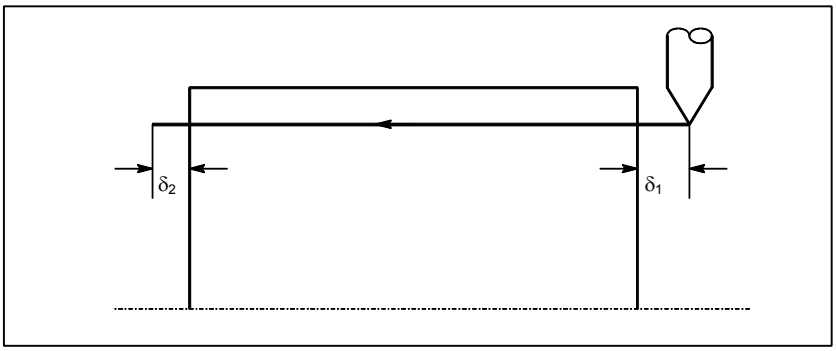


Fig. E.2 (a) Incorrect threaded portion

Explanation

- How to determine δ_2

$$\delta_2 = \frac{LR}{1800^*}(mm)$$

R : Spindle speed (min^{-1})
L : Thread lead (mm)
* When time constant T_1 of the servo system is 0.033 s.

- How to determine δ_1

$$\delta_1 = \frac{LR}{1800^*}(- 1 - \ln a) \ (mm)$$

$$= \delta_2(- 1 - \ln a)(mm)$$

R : Spindle speed (min^{-1})

L : Thread lead (mm)

* When time constant T_1 of the servo system is 0.033 s.

Following a is a permitted value of thread.

a	-1-lna
0.005	4.298
0.01	3.605
0.015	3.200
0.02	2.912

Example

R=350min^{-1}

L=1mm

a=0.01

then

$$\delta_2 = \frac{350 \times 1}{1800} = 0.194(mm)$$

$$\delta_1 = \delta_2 \times 3.605 = 0.701(mm)$$

Reference

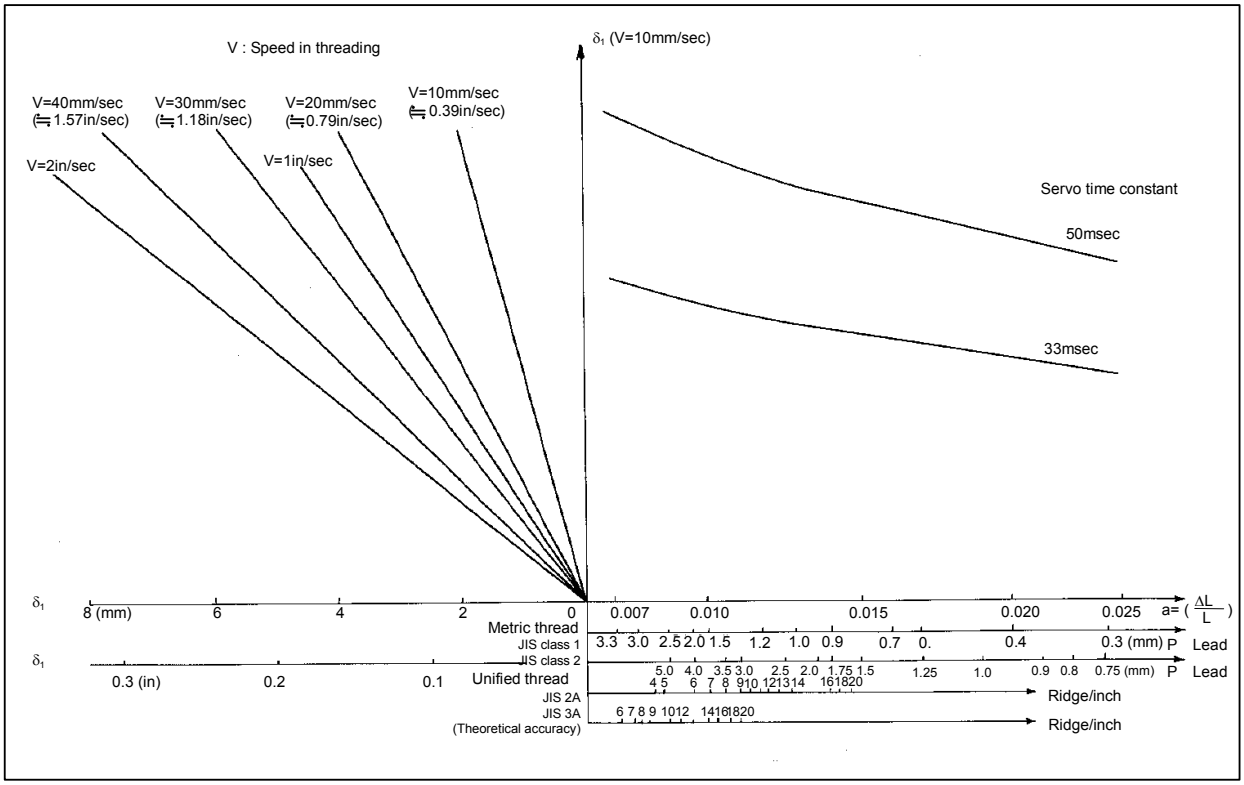


Fig. E.2 (b) Nomograph for obtaining approach distance δ_1

E.3 TOOL PATH AT CORNER

When servo system delay (by exponential acceleration/deceleration at cutting or caused by the positioning system when a servo motor is used) is accompanied by cornering, a slight deviation is produced between the tool path (tool center path) and the programmed path as shown in Fig. E.3 (a).
Time constant T_1 of the exponential acceleration/deceleration is fixed to 0.

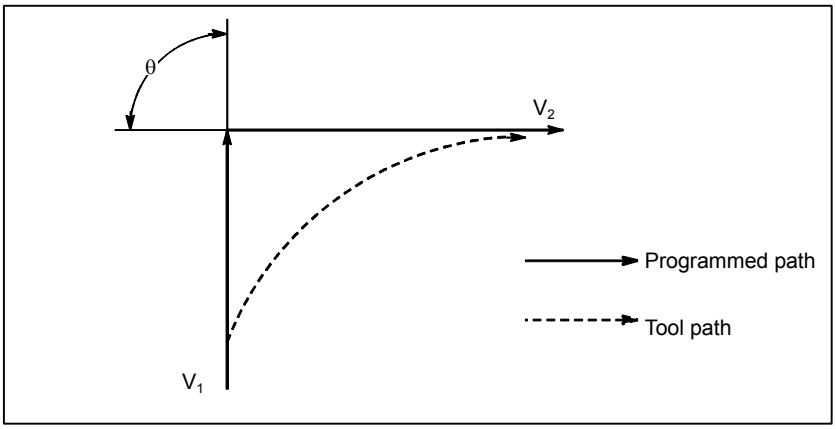


Fig. E.3 (a) Slight deviation between the tool path and the programmed path

This tool path is determined by the following parameters:

- Feedrate (V_1, V_2)
- Corner angle (θ)
- Exponential acceleration / deceleration time constant (T_1) at cutting ($T_1 = 0$)
- Presence or absence of buffer register.

The above parameters are used to theoretically analyze the tool path and above tool path is drawn with the parameter which is set as an example.
When actually programming, the above items must be considered and programming must be performed carefully so that the shape of the workpiece is within the desired precision.
In other words, when the shape of the workpiece is not within the theoretical precision, the commands of the next block must not be read until the specified feedrate becomes zero. The dwell function is then used to stop the machine for the appropriate period.

Explanation

- Analysis

The tool path shown in Fig. E.3 (b) is analyzed based on the following conditions:

- Feedrate is constant at both blocks before and after cornering.
- The controller has a buffer register. (The error differs with the reading speed of the tape reader, number of characters of the next block, etc.)

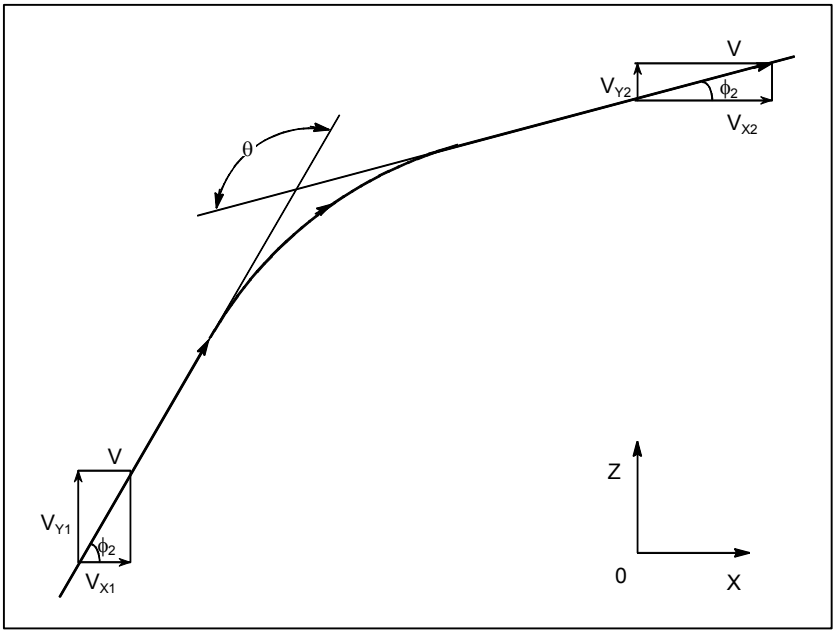


Fig. E.3 (b) Example of tool path

- Description of conditions and symbols

$V_{X1} = V\cos\phi_1$
$V_{Y1} = V\sin\phi_1$
$V_{X2} = V\cos\phi_2$
$V_{Y2} = V\sin\phi 2$

V : Feedrate at both blocks before and after cornering
V_{X1} : X-axis component of feedrate of preceding block
V_{Y1} : Y-axis component of feedrate of preceding block
V_{X2} : X-axis component of feedrate of following block
V_{Y2} : Y-axis component of feedrate of following block
θ : Corner angle
ϕ_1 : Angle formed by specified path direction of preceding block and X-axis
ϕ_2 : Angle formed by specified path direction of following block and X-axis

- Initial value calculation

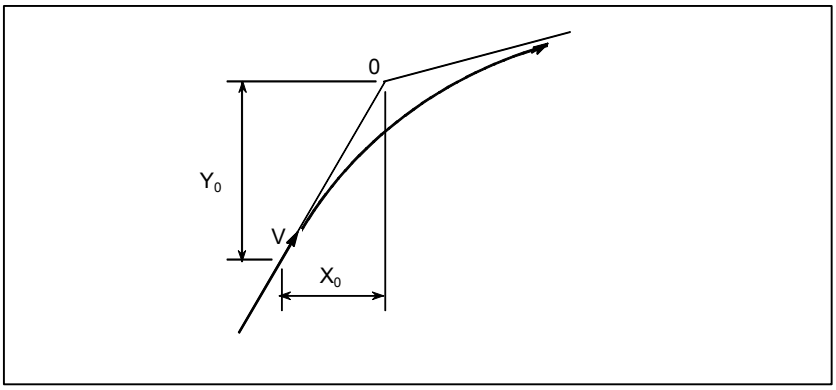


Fig. E.3 (c) Initial value

The initial value when cornering begins, that is, the X and Y coordinates at the end of command distribution by the controller, is determined by the feedrate and the positioning system time constant of the servo motor.

$$X_0 = V_{X1}(T_1 + T_2)$$

$$Y_0 = V_{Y1}(T_1 + T_2)$$

T_1: Exponential acceleration/deceleration time constant. (T=0)
T_2: Time constant of positioning system (Inverse of position loop gain)

- Analysis of corner tool path

The equations below represent the feedrate for the corner section in X-axis direction and Y-axis direction.

$$V_X^{(t)} = (V_{X2} - V_{X1})[1 - \frac{V_{X1}}{T_1 - T_2}\{T_1 \exp(-\frac{t}{T_1}) - T_2 \exp(-\frac{t}{T_2})\} + V_{X1}]$$

$$= V_{X2}[1 - \frac{V_{X1}}{T_1 - T_2}\{T_1 \exp(-\frac{t}{T_1}) - T_2 \exp(-\frac{t}{T_2})\}]$$

$$V_Y^{(t)} = \frac{V_{Y1} - V_{Y2}}{T_1 - T_2}\{T_1 \exp(-\frac{t}{T_1}) - T_2 \exp(-\frac{t}{T_2})\} + V_{Y2}$$

Therefore, the coordinates of the tool path at time t are calculated from the following equations:

$$X(t) = \int_0^1 V_X(t)dt - X_0$$

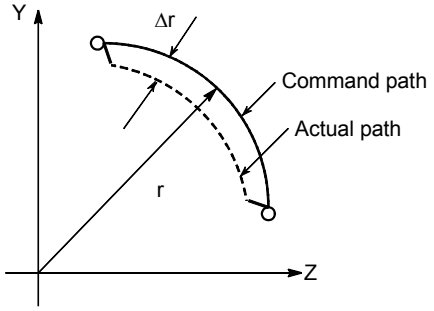
$$= \frac{V_{X2} - V_{X1}}{T_1 - T_2}\{T_1^2 \exp(-\frac{t}{T_1}) - T_2^2 \exp(-\frac{t}{T_2})\} - V_{X2}(T_1 + T_2 - t)$$

$$Y(t) = \int_0^1 V_Y(t)dt - Y_0$$

$$= \frac{V_{Y2} - V_{Y1}}{T_1 - T_2}\{T_1^2 \exp(-\frac{t}{T_1}) - T_2^2 \exp(-\frac{t}{T_2})\} - V_{Y2}(T_1 + T_2 - t)$$

E.4 RADIUS DIRECTION ERROR AT CIRCLE CUTTING

When a servo motor is used, the positioning system causes an error between input commands and output results. Since the tool advances along the specified segment, an error is not produced in linear interpolation. In circular interpolation, however, radial errors may be produced, sepecially for circular cutting at high speeds.
This error can be obtained as follows:



$$\Delta r = \frac{1}{2}(T_1^2 + T_2^2(1-\alpha^2))\frac{V^2}{r} \quad \ldots\ldots (1)$$

Δr: Maximum radius error (mm)
v : Feedrate (mm/sec)
r : Circle radius (mm)
T_1: Exponential acceleration/deceleration time constantat cutting (sec) (T=0)
T_2: Time constant of positoning system (sec)
(Inverse of positon loop gain)
α : Feed forward coefficient (%)

In the case of bell-shaped acceleration/deceleration and linear acceleration/deceleration after cutting feed interpolation, an approximation of this radius error can be obtained with the following expression:

Linear acceleration/deceleration after cutting feed interpolation

$$\Delta r = (\frac{1}{24}T_1^2 + \frac{1}{2}T_2^2(1-\alpha^2))\frac{V^2}{r}$$

Bell-shaped acceleration/deceleration after cutting feed interpolation

$$\Delta r = (\frac{1}{48}T_1^2 + \frac{1}{2}T_2^2(1-\alpha^2))\frac{V^2}{r}$$

Thus, the radius error in the case of bell-shaped acceleration/deceleration and linear acceleration/deceleration after interpolation is smaller than in case of exponential acceleration/deceleration by a factor of 12, excluding any error caused by a servo loop time constant.

Since the machining radius r (mm) and allowable error Δr (mm) of the workpiece is given in actual machining, the allowable limit feedrate v (mm /sec) is determined by equation (1).
Since the acceleration/deceleration time constant at cutting which is set by this equipment varies with the machine tool, refer to the manual issued by the machine tool builder.

F SETTINGS AT POWER-ON, IN THE CLEAR STATE, OR IN THE RESET STATE

Either the clear state or reset state is entered during a reset is set by bit 6 (CLR) of parameter No. 3402 (0: reset state/1: clear state).
The symbols in the figure below have the following meanings.

○ : The setting remains unchanged or the operation continues.
× : The setting is deleted or the operation stops.

Item		Power-on	Clear state	Reset state
Settings	Offset	○	○	○
	Set data	○	○	○
	Parameter	○	○	○
Various data items	Program in memory	○	○	○
	Information of the block read in advance during automatic operation	×	×	×
	N (sequence number)	× (Note 7)	× (Notes 1, 2, 7)	○ (Note 1, 2)
	One-shot G code	×	×	×
	Modal G code	Initial value (Note 3) G20 and G21 are set to the state at the previous power-off.	Initial value (Note 3) G20 and G21 are not changed (Note 2).	○ (Note 2 and 6)
	F	0	0	○ (Note 2)
	S, T, M	×	○ (Note 2)	○ (Note 2)
	K (count specification)	×	×	×
Coordinate	Workpiece coordinates	0 (Note 4)	○	○
Operation during execution	Movement	×	×	×
	Dwell	×	×	×
	M, S, T code output	×	×	×
	Tool position compensation	×	See "Tool Position Compensation at a Reset".	See "Tool Position Compensation at a Reset".
	Tool length compensation	×	Depending on the setting of bit 6 (LVK) of parameter No. 5003	Depending on the setting of bit 6 (LVK) of parameter No. 5003
	Tool radius/tool nose radius compensation	×	×	×
	Storing a called subprogram	×	× (Note 5)	×

Item		Power-on	Clear state	Reset state
Output signals	CNC alarm signal AL	"0"(when no alarm cause is present)	"0" (when no alarm cause is present)	"0" (when no alarm cause is present)
	Reference position return completion signal ZPx	×	○ (or × for emergency stop)	○(or × for emergency stop)
	S, T, B codes	×	○	○
	M code	×	×	×
	M, S, T, B strobe signals	×	×	×
	Spindle rotation signal (S analog signal)	×	○	○
	CNC ready signal MA	"1"	○	○
	Servo ready signal SA	"1" (when a servo alarm does not occur)	"1" (when a servo alarm does not occur)	"1" (when a servo alarm does not occur)
	Cycle start signal STL	×	×	×
	Feed hold signal SPL	×	×	×

Tool position compensation at a reset

○ : Cancelled.

× : Not canceled.

Compensation method		Bit 3 (LVC) of parameter No. 5006 and bit 7 (TGC) of parameter No. 5003			
		LVC=0 TGC=0	LVC=1 TGC=0	LVC=0 TGC=1	LVC=1 TGC=1
Tool movement	Wear compensation	×	○ (Axis movement)	×	○ (Axis movement)
	Geometry compensation				
Coordinate shift	Wear compensation	×	○	×	○
	Geometry compensation	×	×	○	○

NOTE

1 When the beginning position is found, the main program number is displayed.
2 If a reset is made during execution of a block, the states of the modal G code and modal address (such as N, F, S, T, or M) specified in the block are not reflected. The modal information specified in the previous and preceding blocks is kept.
3 The initial value of the modal G code depends on the following parameters. For details, refer to the parameter manual (B-64490EN).
- Bit 0 (G01) of parameter No. 3402
- Bit 1 (G18) of parameter No. 3402
- Bit 2 (G19) of parameter No. 3402
- Bit 3 (G91) of parameter No. 3402
- Bit 7 (G23) of parameter No. 3402
- Parameters No. 3406 to 3409

4 For detection of an absolute position, the value is generated from machine coordinates and a workpiece origin offset.
5 When a reset is made during execution of a subprogram, a return to the main program is performed. Execution at a midpoint in the subprogram is impossible.

NOTE

6 When one of the following two settings, which hold the modal G code in group 1 by a reset is set:
- Reset state (bit 6 of parameter No. 3402 is 0)
- Clear state (bit 1 of parameter No. 3402 is 1) and the modal G code in group 1 is held at a time of a reset (bit 1 of parameter No. 3406 is 1),

if a reset is made during execution of one of the following canned cycles, which cause cycle operation, the mode of the modal G code in group 1 is changed to the G01 mode.
- G90 : Outer surface/inner surface turning cycle
- G92 : Threading cycle
- G94 : Edge turning cycle

7 When bit 0 (SEK) of parameter No. 11353 is set to 1, sequence number N can be held even at power-on or in the clear state.

G CHARACTER-TO-CODES CORRESPONDENCE TABLE

Appendix G, "CHARACTER-TO-CODES CORRESPONDENCE TABLE", consists of the following sections:

G.1 CHARACTER-TO-CODES CORRESPONDENCE TABLE 2177
G.2 FANUC DOUBLE-BYTE CHARACTER CODE TABLE.....2178

G.1 CHARACTER-TO-CODES CORRESPONDENCE TABLE

Character	Code	Comment	Character	Code	Comment
A	065		6	054	
B	066		7	055	
C	067		8	056	
D	068		9	057	
E	069			032	Space
F	070		!	033	Exclamation mark
G	071		“	034	Quotation mark
H	072		#	035	Sharp
I	073		$	036	Dollar sign
J	074		%	037	Percent
K	075		&	038	Ampersand
L	076		‘	039	Apostrophe
M	077		(	040	Left parenthesis
N	078		)	041	Right parenthesis
O	079		*	042	Asterisk
P	080		+	043	Plus sign
Q	081		,	044	Comma
R	082		-	045	Minus sign
S	083		.	046	Period
T	084		/	047	Slash
U	085		:	058	Colon
V	086		;	059	Semicolon
W	087		<	060	Left angle bracket
X	088		=	061	Equal sign
Y	089		>	062	Right angle bracket
Z	090		?	063	Question mark
0	048		@	064	At mark
1	049		[	091	Left square bracket
2	050		]	093	Right square bracket
3	051		^	094	
4	052		_	095	Underscore
5	053				

G.2 FANUC DOUBLE-BYTE CHARACTER CODE TABLE

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
0200	ぁ	け	ち	ぱ	め	材	面	号		回	領	源	規	設	仮	無	対
0220	あ	げ	ぢ	ひ	も	を	最	仕		転	域	投	除	定	想	視	相
0240	ぃ	こ	っ	び	ゃ	ん	小	上		数	診	入	隅	一	副	器	座
0260	い	ご	つ	ぴ	や	種	内	込		位	断	間	取	覧	行	原	標
0280	ぅ	さ	づ	ふ	ゅ	類	大	点		置	操	分	単	表	挿	登	示
02A0	う	ざ	て	ぶ	ゆ	棒	加	方		決	作	秒	補	部	消	録	名
02C0	ぇ	し	で	ぷ	ょ	穴	工	向		直	手	自	能	炭	去	再	歯
02E0	え	じ	と	へ	よ	成	切	速		線	引	運	独	合	山	処	変
0300	ぉ	す	ど	べ	ら	形	削	度		時	機	負	終	金	高	理	呼
0320	お	ず	な	ぺ	り	質	倣	送		円	械	荷	了	鋼	準	描	推
0340	か	せ	に	ほ	る	寸	正	量		反	残	実	記	超	備	画	馬
0360	が	ぜ	ぬ	ぼ	れ	法	途	開		現	移	使	角	硬	完	過	力
0380	き	そ	ね	ぽ	ろ	外	中	始		在	動	用	溝	先	後	容	系
03A0	ぎ	ぞ	の	ま	ゎ	径	荒	深		指	次	寿	刃	付	弧	編	選
03C0	く	た	は	み	わ	長	具	主		令	早	命	幅	摩	助	集	達
03E0	ぐ	だ	ば	む	素	端	番	軸		値	電	新	広	耗	択	未	閉

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
0400	禁	復	帰	書	個	桁	稼	由	両	半	逃	底	逆	下	空	四
0420	触	平	代	辺	格	子	周	心	本	群	停	止	巾	微	状	路
0440	範	囲	倍	率	注	側	特	殊	距	離	連	続	増	隔	件	初
0460	期	条	経	握	圧	扱	陰	隠	右	押	横	黄	億	屋	化	何
0480	絵	階	概	該	巻	換	気	起	軌	技	疑	供	共	境	強	教
04A0	掘	繰	係	傾	型	検	権	研	肩	見	験	元	弦	滅	孔	巧
04C0	控	更	校	構	根	左	差	雑	参	散	産	算	治	耳	式	失
04E0	修	十	従	勝	商	少	尚	昇	植	色	食	伸	信	侵	振	浸
0500	真	暗	以	意	異	影	鋭	越	価	可	科	果	箇	課	各	拡
0520	核	学	掛	漢	簡	観	関	含	却	客	休	急	業	曲	均	筋
0540	継	計	軽	言	限	互	降	採	済	細	姿	思	写	射	斜	者
0560	車	借	縦	重	出	述	術	渉	照	省	章	証	象	身	進	人
0580	図	違	印	沿	遠	央	奥	往	応	会	解	改	割	活	願	基
05A0	奇	寄	岐	既	近	区	矩	駆	偶	旧	求	球	究	級	欠	結
05C0	口	語	誤	交	厚	項	刻	告	黒	財	策	糸	試	資	事	持
05E0	似	釈	弱	受	収	純	順	所	序	剰	場	常	飾	水	錐	据
0600	制	整	製	前	全	然	則	属	即	他	多	存	谷	探	短	徴
0620	鎖	調	頂	鉄	添	頭	同	導	道	熱	年	濃	箱	発	抜	伴
0640	必	百	複	物	文	聞	併	忘	末	密	有	余	与	裏	立	略
0660	青	席	石	積	赤	接	折	粗	創	双	捜	太	打	体	待	態
0680	替	段	知	地	致	遅	追	通	伝	得	読	凸	凹	突	鈍	敗
06A0	杯	背	配	品	不	布	並	頁	別	片	返	勉	弁	保	明	滅
06C0	木	目	歪	揺	様	溶	要	抑	良	輪	和	話	枠	節	説	絶
06E0	千	専	浅	旋	総	走	退	台	第	題	卓	室	着	柱	鋳	丁
0700	低	訂	肉	日	白	薄	比	皮	被	非	美	普	伏	歩	包	門
0720	問	絡	列	万	利	訳	礼	乱	放	枚	約	練	油	劣	例	郭
0740	戻	冷	垂	緑	紫	許	測	精	効	→	↗	↑	↖	←	↙	↓
0760	↘	↻	↻	⌒	⌒	■	板	予	〃	家	装	管	粉	等	▿	▽
0780	▽	▽	貫	安	⌒	β	程	抗	張	任	破	損	御	足	守	般
07A0	納	義	丸	汎	α	毎	当	的	詳	鳥	適	論	額	縁	温	給
07C0	界	混	監	締	固	己	称	樹	脂	料	落	確	認	報	排	性
07E0	生	績	判	搬	護	θ	島	壁	↪	⌭	↪	⌦	🕮	⌭	〕〔	■■

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
0800	阿	哀	愛	挨	逢	悪	旭	宛	案	闇	鞍	伊	依	偉	委	威
0820	慰	易	為	維	緯	胃	衣	遺	医	井	育	一	稲	員	因	飲
0840	院	羽	雨	渦	唄	浦	瓜	噂	雲	営	映	栄	永	泳	洩	英
0860	衛	液	益	駅	園	延	援	演	炎	煙	鉛	塩	汚	王	岡	沖
0880	憶	腋	牡	乙	恩	穏	音	佳	夏	暇	架	歌	河	火	花	華
08A0	菓	貨	我	牙	芽	賀	雅	介	壊	廻	快	怪	海	灰	皆	貝
08C0	劾	害	慨	街	垣	殻	獲	覚	較	革	楽	笠	括	滑	株	刈
08E0	乾	寒	刊	勧	喚	官	寛	干	幹	感	慣	敢	歓	汗	環	甘
0900	看	緩	缶	肝	還	鑑	閑	陥	韓	館	岸	眼	岩	顔	企	危
0920	喜	希	幾	揮	机	旗	棄	毅	祈	季	稀	徽	輝	騎	擬	犠
0940	議	菊	詰	脚	丘	久	及	吸	宮	弓	救	泣	牛	居	巨	拒
0960	挙	虚	魚	亨	享	京	競	協	叫	挟	橋	況	狭	胸	興	郷
0980	鏡	響	驚	凝	局	極	玉	勤	錦	琴	銀	九	句	苦	馳	屑
09A0	屈	熊	君	訓	軍	郡	刑	兄	契	掲	敬	景	茎	警	芸	迎
09C0	劇	激	隙	潔	血	月	倹	健	兼	券	剣	圏	堅	建	憲	拳
09E0	犬	献	絹	県	謙	軒	鍵	険	幻	古	庫	戸	故	湖	狐	誇
0A00	顧	五	午	侯	候	光	公	勾	喉	好	孝	幸	康	弘	拘	攻
0A20	江	港	甲	稿	絞	綱	考	肯	衡	講	購	郊	鉱	香	剛	克
0A40	国	穀	酷	腰	骨	此	頃	今	困	婚	査	砂	債	妻	彩	才
0A60	災	砕	祭	菜	裁	載	際	剤	罪	坂	阪	咲	昨	索	錯	桜
0A80	冊	刷	察	撮	擦	札	殺	皿	三	撤	讃	賛	酸	伺	刺	史
0AA0	士	姉	市	師	支	枝	死	私	紙	詞	詩	字	寺	磁	辞	七
0AC0	湿	芝	縞	捨	煮	社	謝	尺	若	酒	首	授	需	秋	習	臭
0AE0	舟	週	住	柔	宿	祝	縮	熟	春	瞬	盾	巡	暑	女	傷	唱
0B00	奨	将	床	承	招	昭	焼	焦	笑	紹	衝	賞	障	乗	城	情
0B20	浄	蒸	錠	職	唇	寝	審	森	申	神	芯	親	辛	針	震	尽
0B40	陣	須	酢	吹	粋	遂	杉	裾	澄	世	是	勢	征	政	星	晴
0B60	清	盛	聖	声	西	誓	請	静	税	昔	析	籍	責	跡	雪	舌
0B80	占	宣	尖	川	戦	扇	栓	泉	洗	染	潜	船	銑	鮮	善	組
0BA0	訴	倉	層	掃	巣	争	窓	草	騒	像	臓	蔵	贈	造	促	息
0BC0	束	俗	卒	其	揃	尊	村	詑	堕	妥	耐	帯	怠	滞	袋	貸
0BE0	隊	滝	宅	拓	濯	託	濁	奪	脱	棚	誰	嘆	担	淡	団	弾

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
0C00	暖	男	談	池	築	畜	竹	筑	秩	茶	昼	虫	駐	貯	帳	庁
0C20	彫	挑	廷	町	脹	腸	跳	沈	珍	賃	塁	痛	塚	爪	吊	釣
0C40	庭	廷	徒	釘	泥	摘	滴	笛	典	天	展	店	貼	殿	田	吐
0C60	塗	徒	統	砥	努	土	怒	倒	冬	凍	刀	島	東	湯	灯	答
0C80	筒	統	鍋	藤	討	踏	透	働	堂	胴	銅	峠	徳	毒	届	曇
0CA0	謎	鍋	波	南	軟	難	二	句	乳	尿	念	燃	粘	悩	脳	農
0CC0	把	波	派	廃	拝	肺	買	売	博	拍	泊	舶	麦	肌	畑	八
0CE0	罰	版	犯	班	繁	販	飯	盤	否	彼	悲	扉	批	疲	秘	肥
0D00	費	避	飛	尾	鼻	菱	筆	俵	氷	票	評	病	浜	貧	敏	夫
0D20	婦	富	怖	浮	父	符	腐	武	舞	封	風	服	福	腹	払	沸
0D40	噴	憤	簿	紛	丙	兵	幣	柄	米	壁	癖	偏	便	捕	募	墓
0D60	母	簿	房	崩	捧	泡	胞	芳	訪	豊	飽	亡	傍	剖	妨	帽
0D80	忙	房	膜	望	紡	肪	膨	防	北	僕	撲	釦	没	翻	磨	魔
0DA0	幕	膜	茂	満	味	魅	脈	妙	民	務	夢	矛	迷	鳴	免	綿
0DC0	模	茂	友	盲	網	黙	紋	冶	夜	野	矢	役	薬	躍	諭	輸
0DE0	優	友	遊	郵	融	誉	預	幼	揚	曜	洋	葉	陽	養	浴	翼
0E00	螺	来	頼	欄	陸	律	流	留	粒	旅	療	稜	林	臨	隣	涙
0E20	累	励	鈴	暦	歴	烈	裂	労	漏	老	六	脇	惑	詫	湾	腕
0E40	斡	椅	菱	宇	嘘	閲	宴	欧	懐	拐	涯	獲	闇	潟	渇	冠
0E60	患	汽	貴	鬼	偽	戯	欺	喫	窮	糾	拠	漁	恐	狂	脅	仰
0E80	緊	愚	遇	靴	啓	慶	憩	携	撃	傑	嫌	懸	厳	雇	娯	洪
0EA0	紅	耕	航	貢	挫	催	栽	崎	柵	挨	傘	志	施	旨	至	誌
0EC0	識	狩	趣	就	秀	衆	襲	蹴	充	渋	緒	署	諸	叙	掌	訟
0EE0	鐘	壌	織	紳	酔	瀬	誠	繊	漸	繕	塑	礎	阻	奏	族	惰
0F00	戴	諾	叩	旦	誕	恥	仲	宙	忠	抽	兆	懲	抵	敵	撤	党
0F20	盗	糖	陶	闘	督	馴	覇	媒	爆	縛	髪	閥	泌	匹	府	敷
0F40	仏	慕	縫	乏	霧	盟	勇	誘	踊	裸	雷	卵	里	隆	慮	虜
0F60	寮	塁	隷	霊	恋	浪	郎	功	坑	々	〆	〒	㍉	㍊	㍍	㍗
0F80	`	a	b	c	d	e	f	g	h	i	j	k	l	m	n	o	p	q	r	s	t	u	v	w	x	y	z	{	;	}	~	▨
0FA0	Б	Г	Д	Ж	З	И	Й	К	Л	М	П	У	Ф	Ц	Ч	Ш	Щ	Ъ	Ы	Ь	Э	Ю	Я		?	㍻	㍼	㍽	㍾
0FC0	À	Á	Ã	Â	Æ	Ç	È	É	Ê	Ë	Ì	Í	Î	Ï	Ò	Ó	Ô	Õ	Œ	Ø	Ù	Ú	Û	Ÿ	ß	à	á	â	ã	ä	å	æ
0FE0	Å	Ä	Ö	Ü	Ñ	¿	ç	è	é	ê	ë	ì	í	î	ï	ñ	ò	ó	ô	õ	ö	œ	ø	ù	ú	û	ü	ÿ	¡			

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
1000	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P
1020	Q	R	S	T	U	V	W	X	Y	Z	a	b	c	d	e	f
1040	g	h	i	j	k	l	m	n	o	p	q	r	s	t	u	v
1060	w	x	y	z	〃				γ	ε	μ	π	φ	ω	Δ	Σ
1080	Ω								ガ	ギ	グ	ゲ	ゴ	ザ	ジ	ズ	ゼ	ゾ	ダ	ヂ	ヅ	デ	ド
10A0	バ	ビ	ブ	ベ	ボ	パ	ピ	プ	ペ	ポ	ヴ	ヵ	ヶ	📖	ァ	ア	ィ	イ	ゥ	ウ	ェ	エ
10C0	ォ	オ	カ	ガ	キ	ギ	ク	グ	ケ	ゲ	コ	ゴ	サ	ザ	シ	ジ
10E0	ス	ズ	セ	ゼ	ソ	ゾ	タ	ダ	チ	ヂ	ッ	ツ	ヅ	テ	デ	ト
1100	ド	ナ	ニ	ヌ	ネ	ノ	ハ	バ	パ	ヒ	ビ	ピ	フ	ブ	プ	ヘ
1120	ベ	ペ	ホ	ボ	ポ	マ	ミ	ム	メ	モ	ヤ	ャ	ュ	ュ	ョ	ヨ
1140	ラ	リ	ル	レ	ロ	ヮ	ワ	ヲ	ン	ヴ	ヵ	ヶ	゛	゜	🖐	🖩
1160		、	。	,	.	・	:	;	?	!	゛	°	´	、	¨	^
1180	—	￣	、	ゝ	ゞ	〃	仝	々	〆	(	)	ー	—	-	／	＼
11A0	～	‖	｜	…	‥	‘	’	“	”	〔	〕	〔	〕	[	]	{
11C0	}	〈	〉	《	》	「	」	『	』	【	】	+	−	±	×	÷
11E0	＝	≠	<	>	≦	≧	∞	∴	♂	♀	°	′	″	℃	¥	$
1200	¢	£	%	＃	&	*	@	§	☆	★	○	●	◎	◇	◆	□
1220	■	△	▲	▽	▼	※	〒	→	←	↑	↓	=	〒	🖐	🖐	⇔
1240	1/1	2/2	3/3	4/4	5/5	6/6	〼	〔〕	mm	cm	km	cm²	m²	km²	cm³	m³
1260	mg	kg	cc	dl	ℓ	kl	ms	μs	ns	HP	ps	Hz	㈱	©		
1280	亜	芦	尉	壱	逸	芋	姻	韻	詠	疫	悦	謁	猿	殴	翁	虞	且	朽	渓
12A0	卸	嫁	禍	彦	蚊	餓	悔	塊	戒	嚇	岳	樫	喝	褐	虐	軈	恵
12C0	勘	堪	棺	款	憾	艦	頑	忌	紀	飢	棋	宜	儀	吉	恵	朽	渓
12E0	凶	峡	恭	矯	暁	斤	桐	菌	謹	襟	吟	隈	勲	薫
1300	蛍	鶏	鯨	遣	賢	繭	顕	玄	孤	枯	鼓	呉	悟	碁	后	恒	宰	諮	爵	淑	肖	冗	炊
1320	皇	慌	酵	拷	蚕	獄	昆	恨	紺	魂	墾	懇	佐	唆	詐	宰	諮	爵	淑	肖	冗	炊
1340	斎	歳	搾	桟	霊	惨	暫	氏	司	社	肢	嗣	飼	雌	賜	酌	叔	匠	丈	尋
1360	児	侍	滋	慈	囚	疾	執	漆	舎	赦	遮	邪	蛇	勺	酌	叔	匠	丈	尋
1380	寂	朱	珠	儒	准	州	宗	拾	愁	酬	醜	汁	銃	獣	叔	匠	丈	尋
13A0	粛	塾	俊	旬	松	殉	循	潤	遵	庶	如	徐	升	召	匠	丈	尋
13C0	償	抄	昌	晶	殖	沼	宵	症	祥	硝	粧	詔	彰	礁	丈	尋
13E0	畳	嬢	譲	醸	嘱	辱	臣	娠	慎	薪	仁	迅	甚	尋

	00	02	04	06	08	0A	0C	0E	10	12	14	16	18	1A	1C	1E
1400	婿	租	賊	畜	坪	渡	屯	漠	漂	陛	貿	岬	憂	涼	楼	
1420	逝	祖	憎	稚	漬	斗	篤	迫	姫	墳	冒	漫	雄	硫	廊	郁
1440	牲	禅	霜	痴	津	澱	匿	伯	罷	雰	某	慢	裕	竜	朗	浩
1460	斉	薦	藻	壇	鎮	徹	童	賠	碑	覆	坊	抹	猶	柳	露	槻
1480	姓	遷	燥	鍛	陳	哲	洞	陪	卑	侮	褒	又	悠	履	炉	靖
14A0	畝	銭	槽	胆	朕	送	騰	培	披	譜	砲	埋	幽	痢	錬	靱
14C0	菅	践	遭	丹	勅	提	豆	梅	妃	附	俸	妹	唯	吏	廉	鐵
14E0	崇	仙	僧	但	聴	偵	謄	輩	蛮	賦	抱	麻	癒	濫	麗	級
1500	帥	脆	措	孫	逐	呈	奴	豚	鉢	苗	塀	謀	眠	羊	猟	賄
1520	衰	夕	疎	駄	窒	弟	桃	尼	伐	猫	弊	朴	娘	庸	陵	國
1540	睡	斥	壮	舵	嫡	邸	悼	弐	帆	賓	遍	牧	銘	窯	僚	揺
1560	穂	隻	荘	胎	衷	亭	棟	妊	畔	頻	舗	墨	妄	擁	糧	條
1580	錘	惜	桑	泰	著	貞	痘	忍	煩	瓶	暮	堀	猛	謡	厘	櫻
15A0	随	拙	曹	逮	弔	帝	唐	寧	頒	扶	邦	奔	匁	翌	倫	澤
15C0	髄	窃	喪	濯	眺	艇	塔	婆	藩	赴	奉	凡	厄	羅	零	濾
15E0	枢	摂	葬	沢	潮	遁	搭	俳	晩	膚	峰	盆	愉	酪	齢	碌

H ALARM LIST

Appendix H, "ALARM LIST", consists of the following items:

(1) Alarms on program and operation (PS alarm) 2184
(2) Background edit alarms (BG alarm) 2184
(3) Communication alarms (SR alarm) 2184
(4) Parameter writing alarm (SW alarm) 2231
(5) Servo alarms (SV alarm) 2231
(6) Overtravel alarms (OT alarm) 2239
(7) Memory file alarms (IO alarm) 2240
(8) Alarms requiring power to be turned off (PW alarm) 2240
(9) Spindle alarms (SP alarm) 2243
(10) Alarm list (serial spindle) 2245
(11) Overheat alarms (OH alarm) 2260
(12) Other alarms (DS alarm) 2260
(13) Malfunction prevention function alarms (IE alarm) 2265

(1) Alarms on program and operation (PS alarm)
(2) Background edit alarms (BG alarm)
(3) Communication alarms (SR alarm)

Alarm numbers are common to all these alarm types.
Depending on the state, an alarm is displayed as in the following examples:
PS"alarm number" Example: PS0003
BG"alarm number" Example: BG0085
SR"alarm number" Example: SR0001

Number	Message	Description
0001	TH ERROR	A TH error was detected during reading from an input device. The read code that caused the TH error and how many statements it is from the block can be verified in the diagnostics screen.
0002	TV ERROR	An error was detected during the single–block TV error. The TV check can be suppressed by setting bit 0 (TVC) of parameter No. 0000 to “0”.
0003	TOO MANY DIGIT	Data entered with more digits than permitted in the NC instruction word. The number of permissible digits varies according to the function and the word.
0004	INVALID BREAK POINT OF WORDS	NC word(s) address + numerical value not in word format. This alarm is also generated when a custom macro does not contain a reserved word, or does not conform to the syntax.
0005	NO DATA AFTER ADDRESS	NC word(s) address + numerical value not in word format. This alarm is also generated when a custom macro does not contain a reserved word, or does not conform to the syntax.
0006	ILLEGAL USE OF MINUS SIGN	A minus sign (–) was specified at an NC instruction word or system variable where no minus signal may be specified.
0007	ILLEGAL USE OF DECIMAL POINT	A decimal point (.) was specified at an address where no decimal point may be specified, or two decimal points were specified.
0009	IMPROPER NC-ADDRESS	An illegal address was specified, or parameter 1020 is not set.

Number	Message	Description
0010	IMPROPER G-CODE	1) An unusable G code is specified. 2) The continuous circle motion-based groove cutting option parameter is not effective. 3) The continuous circle motion-based groove cutting enable signal is "0".
0011	FEED ZERO (COMMAND)	1) The cutting feedrate instructed by an F code has been set to 0. 2) This alarm is also generated if the F code instructed for the S code is set extremely small in a rigid tapping instruction as the tool cannot cut at the programmed lead. 3) During continuous circle motion-based groove cutting, correct Q or F value is not specified or the acceleration clamp value for continuous circle motion in parameter No. 3490 is invalid.
0014	CAN NOT COMMAND G95	A synchronous feed is specified without the option for threading / synchronous feed. Modify the program.
0015	TOO MANY SIMULTANEOUS AXES	A move command was specified for more axes than can be controlled by simultaneous axis control. Either add on the simultaneous axis control extension option, or divide the number of programmed move axes into two blocks.
0020	OVER TOLERANCE OF RADIUS	An arc was specified for which the difference in the radius at the start and end points exceeds the value set in parameter No. 3410. Check arc center codes I, J and K in the program. The tool path when parameter No. 3410 is set to a large value is spiral.
0021	ILLEGAL PLANE SELECT	The plane selection instructions G17 to G19 are in error. Reprogram so that same 3 basic parallel axes are not specified simultaneously. This alarm is also generated when an axis that should not be specified for plane machining is specified, for example, for circular interpolation or involute interpolation. To enable programming of 3 or more axes, the helical interpolation option must be added to each of the relevant axes.
0022	R OR I,J,K COMMAND NOT FOUND	The command for circular interpolation lacks arc radius R or coordinate I, J, or K of the distance between the start point to the center of the arc.
0025	CIRCLE CUT IN RAPID (F0)	F0 (rapid traverse in inverse feed or feed specified by an F code with 1–digit number) was specified during circular interpolation (G02, G03) or involute interpolation (G02.2, G03.2).
0027	NO AXES COMMANDED IN G43/G44	No axis is specified in G43 and G44 blocks for the tool length offset type C. Offset is not canceled but another axis is offset for the tool length offset type C. Multiple axes were specified for the same block when the tool length compensation type is C.
0029	ILLEGAL OFFSET VALUE	Illegal offset No.

Number	Message	Description
0030	ILLEGAL OFFSET NUMBER	An illegal offset No. was specified. This alarm is also generated when the tool shape offset No. exceeds the maximum number of tool offset sets in the case of tool offset memory B. In cutting point command, this alarm is also generated when the corner R offset No. ([H code + 2] or [H code + 2 * (parameter No. 11419)]) exceeds the maximum number of tool offset sets in the case of tool offset memory A and B.
0031	ILLEGAL P COMMAND IN G10	The relevant data input or option could not be found for the L No. of G10. No data setting address such as P or R was specified. An address command not concerned with data setting was specified. An address varies with the L No. The sign or decimal point of the specified address is in error, or the specified address is out of range.
0032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.
0033	NO INTERSECTION AT CUTTER COMPENSATION	The intersection cannot be obtained by the intersection calculation in tool radius/tool nose radius compensation. Modify the program.
0034	NO CIRC ALLOWED IN STUP/EXT BLK	In tool radius/tool nose radius compensation, a startup or cancellation is performed in the G02 or G03 mode. Modify the program.
0035	CAN NOT COMMANDED G31	- G31 cannot be specified. This alarm is generated when a G code (such as for tool radius/tool nose radius compensation) of group 07 is not canceled. - A torque limit skip was not specified in a torque limit skip command (G31P98 or P99). Specify the torque limit skip in the PMC window or the like. Or, specify the torque limit override by address Q.
0037	CAN NOT CHANGE PLANE IN G41/G42	The compensation plane G17/G18/G19 was changed during cutter or tool-nose radius compensation. Modify the program.
0038	INTERFERENCE IN CIRCULAR BLOCK	Overcutting will occur in tool radius/tool nose radius compensation because the arc start point or end point coincides with the arc center. Modify the program.
0039	CHF/CNR NOT ALLOWED IN G41,G42	Chamfering or corner R was specified with a start-up, a cancel, or switching between G41 and G42 in G41 and G42 commands (tool radius/tool nose radius compensation). The program may cause overcutting to occur in chamfering or corner R. Modify the program.
0041	INTERFERENCE IN CUTTER COMPENSATION	In tool radius/tool nose radius compensation, excessive cutting may occur. Modify the program.
0042	G45/G48 NOT ALLOWED IN CRC	Tool offset (G45 to G48) is commanded in tool radius compensation or 3-dimensional cutter compensation. Modify the program.
0043	ILLEGAL T-CODE COMMAND	On a system with a DRILL-MATE ATC installed, M06 is not specified in a block that specifies a T code. Alternatively, a T code beyond the allowable range is specified.
0044	G27-G30 NOT ALLOWED IN FIXED CYC	One of G27 to G30 is commanded in canned cycle mode. Modify the program.
0045	ADDRESS Q NOT FOUND (G73/G83)	In a high-speed peck drilling cycle (G73) or peck drilling cycle (G83), the amount of each-time cutting is not specified by address Q, or Q0 is specified. Modify the program.

Number	Message	Description
0046	ILLEGAL REFERENCE RETURN COMMAND	A command for a return to the second, third or fourth reference position is error. (The address P command is in error.) Although an option for a return to the third or fourth reference position was not set, 3 or 4 was specified in address P.
0047	ILLEGAL AXIS SELECT	Two or more parallel axes (in parallel with a basic axis) have been specified upon start-up of 3-dimensional tool compensation or 3-dimensional coordinate conversion.
0048	BASIC 3 AXIS NOT FOUND	Start-up of 3-dimensional tool compensation or 3-dimensional coordinate conversion has been attempted, but the three basic axes used when Xp, Yp, or Zp is omitted are not set in parameter No. 1022.
0049	ILLEGAL COMMAND(G68,G69)	When 3-dimensional coordinate conversion (G68 or G69) was specified, the tool compensation was not canceled. Or, programs of 3-dimensional coordinate conversion (G68, G69) and tool compensation (G43, G44 or G49) were not nested. Or, the 3-dimensional coordinate conversion was specified during the tool length compensation and another tool length compensation was specified.
0050	CHF/CNR NOT ALLOWED IN THRD BLK	Chamfering or corner R is commanded in the thread cutting block. Modify the program.
0051	MISSING MOVE AFTER CNR/CHF	Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program.
0052	CODE IS NOT G01 AFTER CHF/CNR	The block next to the chamfering or corner R block is not G01 (or vertical line). Modify the program.
0053	TOO MANY ADDRESS COMMANDS	In the chamfering and corner R commands, two or more of I, J, K and R are specified.
0054	NO TAPER ALLOWED AFTER CHF/CNR	A block in which chamfering in the specified angle or the corner R was specified includes a taper command. Modify the program.
0055	MISSING MOVE VALUE IN CHF/CNR	In chamfering or corner R block, the move distance is less than chamfer or corner R amount. Modify the program.
0056	NO END POINT & ANGLE IN CHF/CNR	In direct dimension drawing programming, both an end point and an angle were specified in the block next to the block in which only an angle was specified (Aa). Modify the program.
0057	NO SOLUTION OF BLOCK END	Block end point is not calculated correctly in direct dimension drawing programming. Modify the program.
0058	END POINT NOT FOUND	Block end point is not found in direct dimension drawing programming. Modify the program.
0060	SEQUENCE NUMBER NOT FOUND	[External data input/output] The specified number could not be found for program number and sequence number searches. Although input/output of a pot number of tool data or offset input was requested, no tool number was input after power on. The tool data corresponding to the entered tool number could not be found. [External workpiece number search] The program corresponding to the specified workpiece number could not be found. [Program restart] In the program restart sequence number specification, the specified sequence number could not be found.
0061	P OR Q COMMAND IS NOT IN THE MULTIPLE REPETIVE CYCLES BLOCK	Address P or Q is not specified in multiple repetitive cycle (G70, G71, G72, or G73) command.

Number	Message	Description
0062	THE CUTTING AMOUNT IS ILLEGAL IN THE ROUGH CUTTING CYCLE	A zero or a negative value was specified in a multiple repetitive canned rough-cutting cycle (G71 or G72) as the depth of cut.
0063	THE BLOCK OF A SPECIFIED SEQUENCE NUMBER IS NOT FOUND	The sequence number specified by addresses P and Q in multiple repetitive cycle (G70, G71, G72, or G73) command cannot be searched.
0064	THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(FIRST AXES)	In a shape program for the multiple repetitive canned rough-cutting cycle (G71 or G72), the command for the first plane axis was not a monotonous increase or decrease.
0065	G00/G01 IS NOT IN THE FIRST BLOCK OF SHAPE PROGRAM	In the first block of the shape program specified by P of the multiple repetitive canned cycle (G70, G71, G72, or G73), G00 or G01 was not specified.
0066	UNAVAILABLE COMMAND IS IN THE MULTIPLE REPETIVE CYCLES BLOCK	An unavailable command was found in a multiple repetitive canned cycle (G70, G71, G72, or G73) command block.
0067	THE MULTIPLE REPETIVE CYCLES IS NOT IN THE PART PROGRAM STORAGE	A multiple repetitive canned cycle (G70, G71, G72, or G73) command is not registered in a tape memory area.
0069	LAST BLOCK OF SHAPE PROGRAM IS AN ILLEGAL COMMAND	In a shape program in the multiple repetitive canned cycle (G70, G71, G72, or G73), a command for the chamfering or corner R in the last block is terminated in the middle.
0070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.
0071	DATA NOT FOUND	- The address to be searched was not found. - The program with specified program number was not found in program number search. - In the program restart block number specification, the specified block number could not be found. Check the data.
0072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 63 (basic), 125 (option), 200 (option), 400 (option) or 1000 (option). Delete unnecessary programs and execute program registration again.
0073	PROGRAM NUMBER ALREADY IN USE	The commanded program number has already been used. Change the program number or delete unnecessary programs and execute program registration again.
0074	ILLEGAL PROGRAM NUMBER	The program number is other than 1 to 9999. Modify the program number.
0075	PROTECT	An attempt was made to register a program whose number was protected. In program matching, the password for the encoded program was not correct. An attempt was made to select a program being edited in the background as the main program. An attempt was made to call a program being edited in the background as a subprogram.
0076	PROGRAM NOT FOUND	The specified program is not found in the subprogram call, macro call or graphic copy. The M, G, T or S codes are called by a P instruction other than that in an M98, G65, G66, G66.1 or interrupt type custom macro, and a program is called by a No. 2 auxiliary function code. This alarm is also generated when a program is not found by these calls.

Number	Message	Description
0077	TOO MANY SUB,MACRO NESTING	The total number of subprogram and macro calls exceeds the permissible range. Another subprogram call was executed during an external memory subprogram call.
0078	SEQUENCE NUMBER NOT FOUND	The specified sequence No. was not found during sequence number search. The sequence No. specified as the jump destination in GOTO— and M99P— was not found.
0079	PROGRAM NOT MATCH	The program in memory does not match the program stored on tape. Multiple programs cannot be matched continuously when bit 3 of parameter No. 2200 is set to “1”. Set bit 3 of parameter No. 2200 to “0” before executing a match.
0080	G37 MEASURING POSITION REACHED SIGNAL IS NOT PROPERLY INPUT	- For machining center series When the tool length measurement function (G37) is performed, a measuring position reached signal goes 1 in front of the area determined by the ε value specified in parameter No.6254. Alternatively, the signal does not go 1. - For lathe When the automatic tool compensation function (G36, G37) is used, a measuring position reached signals (XAE1, XAE2) does not go 1 within the range determined by the ε value specified in parameters Nos.6254 and 6255.
0081	G37 OFFSET NO. UNASSIGNED	- For machining center series The tool length measurement function (G37) is specified without specifying an H code. Correct the program. - For lathe The automatic tool compensation function (G36, G37) is specified without specifying an T code. Correct the program.
0082	G37 SPECIFIED WITH H CODE	- For machining center series The tool length measurement function (G37) is specified together with an H code in the same block. Correct the program. - For lathe The automatic tool compensation function (G36, G37) is specified together with an T code in the same block. Correct the program.
0083	G37 IMPROPER AXIS COMMAND	- For machining center series An error has been found in axis specification of the tool length measurement function (G37). Alternatively, a move command is specified as an incremental command. Correct the program. - For lathe An error has been found in axis specification of the automatic tool compensation function (G36, G37). Alternatively, a command is specified as an incremental command. Correct the program.
0085	OVERRUN ERROR	The next character was received from the I/O device connected to RS232C interface 1 before it could read a previously received character.

Number	Message	Description
0086	DR OFF	During I/O process by RS232C interface 1, the data set ready input signal of the I/O device (DR) was OFF. Possible causes are an I/O device not turn on, a broken cable, and a defective printed circuit board.
0087	BUFFER OVERFLOW	During a read by RS232C interface 1, although a read stop command was issued, more than 10 characters were input. The I/O device or printed circuit board was defective.
0090	REFERENCE RETURN INCOMPLETE	1. The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return. 2. An attempt was made to set the zero position for the absolute position detector by return to the reference position when it was impossible to set the zero point. Rotate the motor manually at least one turn, and set the zero position of the absolute position detector after turning the CNC and servo amplifier off and then on again.
0091	MANUAL REFERENCE POSITION RETURN IS NOT PERFORMED IN FEED HOLD	Manual return to the reference position cannot be performed when automatic operation is halted. Perform the manual return to the reference position when automatic operation is stopped or reset.
0092	ZERO RETURN CHECK (G27) ERROR	The axis specified in G27 has not returned to zero. Reprogram so that the axis returns to zero.
0094	P TYPE NOT ALLOWED (COORD CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the coordinate system setting operation was performed.) Perform the correct operation according to the OPERATOR'S MANUAL.
0095	P TYPE NOT ALLOWED (EXT OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the external workpiece offset amount changed.) Perform the correct operation according to the OPERATOR'S MANUAL.
0096	P TYPE NOT ALLOWED (WRK OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the workpiece offset amount changed.) Perform the correct operation according to the OPERATOR'S MANUAL.
0097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be directed when the program is restarted. (After power ON, after emergency stop or alarms 0094 to 0097 reset, no automatic operation is performed.) Perform automatic operation.
0098	G28 FOUND IN SEQUENCE RETURN	A command of the program restart was specified without the reference position return operation after power ON or emergency stop, and G28 was found during search. Perform the reference position return.
0099	MDI EXEC NOT ALLOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI.
0109	FORMAT ERROR IN G08	A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.
0110	OVERFLOW :INTEGER	An integer went out of range during arithmetic calculations.
0111	OVERFLOW :FLOATING	A decimal point (floating point number format data) went out of range during arithmetic calculations.
0112	ZERO DIVIDE	An attempt was made to divide by zero in a custom macro.
0113	IMPROPER COMMAND	A function which cannot be used in custom macro is commanded. Modify the program.

Number	Message	Description
0114	ILLEGAL EXPRESSION FORMAT	The format used in an expression in a custom macro statement is in error. The parameter tape format is in error.
0115	VARIABLE NO. OUT OF RANGE	A number that cannot be used for a local variable, common variable, or system variable in a custom macro is specified. In the EGB axis skip function or skip function for flexible synchronous control (G31.8), a non-existent custom macro variable number is specified. Or, the number of custom macro variables used to store skip positions is not sufficient. Alternatively, the header data in high-speed cycle machining is improper. This alarm is issued in the following cases. 1) The header corresponding to the specified call machining cycle number is absent. 2) The value of cycle connection information falls outside the allowable range (0 to 999). 3) The number of data items in the header falls outside the allowable range (1 to 65535). 4) The storage start data variable number of executable data falls outside the allowable ranges (#20000 to #85535/#200000 to #986431/#2000000 to #3999999). 5) The storage end data variable number of executable data falls outside the allowable ranges (#85535/#986431/#3999999). 6) The storage start data variable number of executable data is the same as the variable number used by the header.
0116	WRITE PROTECTED VARIABLE	An attempt was made in a custom macro to use on the left side of an expression a variable that can only be used on the right side of an expression.
0118	TOO MANY BRACKET NESTING	Too many brackets “[ ]” were nested in a custom macro. The nesting level including function brackets is 5.
0119	ARGUMENT VALUE OUT OF RANGE	The value of an argument in a custom macro function is out of range.
0122	TOO MANY MACRO NESTING	Too many macro calls were nested in a custom macro.
0123	ILLEGAL MODE FOR GOTO/WHILE/DO	A GOTO statement or WHILE–DO statement was found in the main program in the MDI or DNC mode.
0124	MISSING END STATEMENT	The END instruction corresponding to the DO instruction was missing in a custom macro.
0125	MACRO STATEMENT FORMAT ERROR	The format used in a macro statement in a custom macro is in error.
0126	ILLEGAL LOOP NUMBER	DO and END Nos. in a custom macro are in error, or exceed the permissible range (valid range: 1 to 3).
0127	DUPLICATE NC,MACRO STATEMENT	An NC statement and macro statement were specified in the same block.
0128	ILLEGAL MACRO SEQUENCE NUMBER	The specified sequence No. could not be found for sequence number search. The sequence No. specified as the jump destination in GOTO-- and M99P-- could not be found.
0129	USE 'G' AS ARGUMENT	Illegal address is commanded in a custom macro call. Please confirm whether the address is valid in the macro call. When using the axis name expansion, please confirm the setting of the parameter (No.11647).
0130	NC AND PMC AXIS ARE CONFLICTED	The NC command and the PMC axis control command were conflicted. Modify the program or ladder.
0136	SPOS AXIS - OTHER AXIS SAME TIME	The spindle positioning axis and another axis are specified in the same block.
0137	M-CODE & MOVE CMD IN SAME BLK.	The spindle positioning axis and another axis are specified in the same block.

Number	Message	Description
0138	SUPERIMPOSED DATA OVERFLOW	The total distribution amount of the CNC and PMC is too large during superimposed control for PMC axis control.
0139	CANNOT CHANGE PMC CONTROL AXIS	The PMC axis was selected for the axis for which the PMC axis is being controlled.
0140	PROGRAM NUMBER ALREADY IN USE	In the background, an attempt was made to select or delete the program being selected in the foreground. Perform the correct operation for the background edition.
0141	CAN NOT COMMAND G51 IN 3-D OFFSET	G51 (Scaling ON) is commanded in the 3-dimensional tool compensation mode. Modify the program.
0142	ILLEGAL SCALE RATE	The scaling rate is 0 times or 10000 times or more. Modify the setting of the scaling rate. (G51P_ … or G51I_J_K_ … or parameter No. 5411 or 5421)
0143	COMMAND DATA OVERFLOW	An overflow occurred in the storage length of the CNC internal data. This alarm is also generated when the result of internal calculation of scaling, coordinate rotation and cylindrical interpolation overflows the data storage. It also is generated during input of the manual intervention amount.
0144	ILLEGAL PLANE SELECTED	The coordinate rotation plane and arc or tool radius·tool nose radius compensation plane must be the same. Modify the program.
0145	ILLEGAL USE OF G12.1/G13.1	The axis No. of plane selection parameter No. 5460 (linear axis) and No. 5461(axis of rotation) in the polar coordinate interpolation mode is out of range (1 to number of controlled axes).
0146	ILLEGAL USE OF G-CODE	The modal G code group contains an illegal G code in the polar coordinate interpolation mode or when a mode was canceled. Only the following G codes are allowed: G40, G50, G69.1 An illegal G code was specified while in the polar coordinate interpolation mode. The following C codes are not allowed: G27, G28, G30, G30.1, G31 to G31.4, G37 to G387.3, G52, G92, G53, G17 to G19, G81 to G89, G68 In the 01 group, G codes other than G01, G02, G03, G02.2 and G03.2 cannot be specified.
0148	SETTING ERROR	Automatic corner override deceleration rate is out of the settable range of judgement angle. Modify the parameters (No.1710 to No.1714).
0149	FORMAT ERROR IN G10L3	In registration (G10L3 to G11) of tool life management data, an address other than Q1, Q2, P1, and P2 or an unusable address was specified.
0150	ILLEGAL LIFE GROUP NUMBER	The tool group number exceeded the maximum allowable value. The tool group number (P after specification of G10 L3;) or the group number given by the tool life management T code in a machining program.
0151	GROUP NOT FOUND AT LIFE DATA	The tool group specified in a machining program is not set in tool life management data.
0152	OVER MAXIMUM TOOL NUMBER	The number of tools registered in one group exceeded the maximum allowable registration tool number.
0153	T-CODE NOT FOUND	In registration of tool life data, a block in which the T code needs to be specified does not include the T code. Alternatively, in tool exchange method D, M06 is specified solely. Modify the program.

Number	Message	Description
0154	NOT USING TOOL IN LIFE GROUP	- For the tool management command H99 or D99 was specified when no tool management data number is assigned to the spindle position. Modify the program. - For the tool life management command The H99 command, D99 command, or the H/D code set by parameters Nos. 13265 and 13266 was specified when no tool belonging to a group is used.
0155	ILLEGAL T-CODE COMMAND	In the machining program, the T code that is present in the block containing M06 does not correspond to the group currently being used. Modify the program.
0156	P/L COMMAND NOT FOUND	The P and L commands are not specified in the beginning of a program for setting a tool group. Modify the program.
0157	TOO MANY TOOL GROUPS	In registration of tool life management data, the group setting command block counts of P (group number) and L (tool life) exceeded the maximum group count.
0158	TOOL LIFE VALUE OUT OF RANGE	The life value that is being set is too large. Change the setting.
0159	ILLEGAL TOOL LIFE DATA	Tool life management data is corrupted for some reason. Register the tool data in the tool group or the tool data in the group again by G10L3; or MDI input.
0160	MISMATCH WAITING M-CODE	A waiting M-code is in error. <1> When different M codes are specified for path 1 and path 2 as waiting M codes without a P command. <2> When the waiting M codes are not identical even though the P commands are identical <3> When the waiting M codes are identical and the P commands are not identical (This occurs when a P command is specified with binary value.) <4> When the number lists in the P commands contain a different number even though the waiting M codes are identical (This occurs when a P command is specified by combining path numbers.) <5> When a waiting M code without a P command (2-path waiting) and a waiting M code with a P command (3-or-more-path waiting) were specified at the same time <6> When a waiting M code without a P command was specified for 3 or more paths.
0161	ILLEGAL P OF WAITING M-CODE	P in a waiting M-code is incorrect. <1> When address P is negative <2> When a P value inappropriate for the system configuration was specified <3> When a waiting M code without a P command (2-path waiting) was specified in the system having 3 or more paths.
0163	ILLEGAL COMMAND IN G68/G69	G68 and G69 are not independently commanded in balance cut. An illegal value is commanded in a balance cut combination (address P).
0169	ILLEGAL TOOL GEOMETRY DATA	Incorrect tool figure data in interference check. Set correct data, or select correct tool figure data.

Number	Message	Description
0175	ILLEGAL G07.1 AXIS	An axis which cannot perform cylindrical interpolation was specified. More than one axis was specified in a G07.1 block. An attempt was made to cancel cylindrical interpolation for an axis that was not in the cylindrical interpolation mode. For the cylindrical interpolation axis, set not “0” but one of 5, 6 or 7 (parallel axis specification) to parameter No. 1022 to instruct the arc with axis of rotation (ROT parameter No. 1006#1 is set to “1” and parameter No. 1260 is set) ON.
0176	ILLEGAL G-CODE USE(G07.1 MODE)	A G code was specified that cannot be specified in the cylindrical interpolation mode. This alarm also is generated when an 01 group G code was in the G00 mode or code G00 was instructed. Cancel the cylindrical interpolation mode before instructing code G00.
0177	CHECK SUM ERROR (G05)	A checksum error occurred.
0178	ILLEGAL COMMAND G05	This alarm is issued in the following cases. 1) The settings of bits 4 to 6 of parameter No.7501 are invalid. 2) G05 was specified in any of the following mode. - Hypothetical axis interpolation (G07) - Cylindrical interpolation (G07.1) - Polar coordinate interpolation (G12.1) - Polar coordinates command (G16) - Spindle speed fluctuation detection (G26) - Tool radius · tool nose radius compensation (G41/G42) - Normal direction control (G41.1/G42.1) - Scaling (G51) - Programmable mirror image (G51.1) - Coordinate system rotation (G68) - Canned cycle (G81 to G89) - Constant surface speed control (G96) - Macro interruption(M96) 3) G05 was specified while the Servo invalidation function and the Pole position detection function are effective at the same time.
0179	PARAM. (NO.7510) SETTING ERROR	The number of controlled axes set by the parameter No. 7510 exceeds the maximum number. Modify the parameter setting value. The distribution of high-speed cycle machining or high-speed binary program operation stopped.
0190	ILLEGAL AXIS SELECTED (G96)	An illegal value was specified in P in a G96 block or parameter No. 5844.
0194	SPINDLE COMMAND IN SYNCHRO-MODE	A Cs contour control mode, spindle positioning command, or rigid tapping mode was specified during the spindle synchronous control mode or spindle command synchronous control mode.
0197	C-AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cs-axis when the Cs contour control switching signal was off.
0199	MACRO WORD UNDEFINED	Undefined macro word was used. Modify the custom macro.
0200	ILLEGAL S CODE COMMAND	In the rigid tap, an S value was out of range or was not specified. The parameters Nos. 5241 to 5243 setting is an S value which can be specified for the rigid tap. Correct the parameters or modify the program.

Number	Message	Description
0201	FEEDRATE NOT FOUND IN RIGID TAP	The command F code for a cutting feedrate is a zero. If the value of F command is much smaller than that of the S command, when a rigid tap command is specified, this alarm is generated. This is because cutting is not possible by the lead specified by the program.
0202	POSITION LSI OVERFLOW	In the rigid tap, spindle distribution value is too large. (System error)
0203	PROGRAM MISS AT RIGID TAPPING	In the rigid tap, position for a rigid M code (M29) or an S command is incorrect. Modify the program.
0204	ILLEGAL AXIS OPERATION	In the rigid tap, an axis movement is specified between the rigid M code (M29) block and G84 (or G74) block. Modify the program.
0205	RIGID MODE DI SIGNAL OFF	Although a rigid M code (M29) is specified in rigid tapping, the rigid mode DI signal (DGN G061.0) is not ON during execution of the G84 (or G74) block. Check the PMC ladder diagram to find the reason why the DI signal is not turned on.
0206	CAN NOT CHANGE PLANE (RIGID TAP)	Plane changeover was instructed in the rigid mode. Modify the program.
0207	RIGID DATA MISMATCH	The specified distance was too short or too long in rigid tapping.
0210	CAN NOT COMMAND M198/M99	1 The execution of an M198 or M99 command was attempted during scheduled operation. Alternatively, the execution of an M198 command was attempted during DNC operation. Modify the program. 2 The execution of an M99 command was attempted by an interrupt macro during pocket machining in a multiple repetitive canned cycle.
0212	ILLEGAL PLANE SELECT	The direct drawing dimensions programming is commanded for the plane other than the Z-X plane. Correct the program.
0213	ILLEGAL COMMAND IN SYNCHRO-MODE	In feed axis synchronization control, the following errors occurred during the synchronous operation. 1) The program issued the move command to the slave axis. 2) The program issued the manual operation (jog feed or incremental feed) to the slave axis. 3) The program issued the automatic reference position return command without specifying the manual reference position return after the power was turned on.
0214	ILLEGAL COMMAND IN SYNCHRO-MODE	Coordinate system is set or tool compensation of the shift type is executed in the synchronous control. Correct the program.
0217	DUPLICATE G51.2(COMMANDS)	G51.2 is further commanded in the G51.2 mode. Modify the program.
0218	NOT FOUND P/Q COMMAND	P or Q is not commanded in the G51.2 block, or the command value is out of the range. Modify the program. For a polygon turning between spindles, more information as to why this alarm occurred is indicated in diagnosis data No. 471.
0219	COMMAND G51.2/G50.2 INDEPENDENTLY	G51.2 and 50.2 were specified in the same block for other commands. Modify the program in another block.
0220	ILLEGAL COMMAND IN SYNCHR-MODE	In the synchronous operation, movement is commanded by the NC program or PMC axis control interface for the synchronous axis. Modify the program or check the PMC ladder.
0221	ILLEGAL COMMAND IN SYNCHR-MODE	Polygon machining synchronous operation and axis control or balance cutting are executed at a time. Modify the program.

Number	Message	Description
0224	ZERO RETURN NOT FINISHED	A reference return has not been performed before the start of automatic operation. (Only when bit 0 (ZRNx) of parameter No. 1005 is 0) Perform a reference position return.
0230	R CODE NOT FOUND	Cut depth R is not specified in the block including G161. Alternatively, the value specified for R is negative. Modify the program.
0231	ILLEGAL FORMAT IN G10 L52	Errors occurred in the specified format at the programmable-parameter input.
0232	TOO MANY HELICAL AXIS COMMAND	Three or more axes were specified as helical axes in the helical interpolation mode. Five or more axes were specified as helical axes in the helical interpolation B mode.
0233	DEVICE BUSY	When an attempt was made to use a unit such as that connected via the RS-232-C interface, other users were using it.
0241	ILLEGAL FORMAT IN G02.2/G03.2	The end point of an involute curve on the currently selected plane, or the center coordinate instruction I, J or K of the corresponding basic circle, or basic circle radius R was not specified.
0242	ILLEGAL COMMAND IN G02.2/G03.2	An illegal value was specified in the involute curve. The coordinate instruction I, J or K of the basic circle on the currently selected plane or the basic circle radius R is “0”, or the start and end points are not inside the basic circle.
0243	OVER TOLERANCE OF END POINT	The end point is not positioned on the involute curve that passes through the start point, and this error exceeds the permissible error limit (parameter No. 5610).
0245	T-CODE NOT ALLOWED IN THIS BLOCK	One of the G codes, G50, G10, G04, G28, G28.2, G29, G30, G30.2, G30.1, and G53, which cannot be specified in the same block as a T code, was specified with a T code.
0247	THE MISTAKE IS FOUND IN THE OUTPUT CODE OF DATA.	When an encrypted program is output, EIA is set for the output code. Specify ISO.
0250	TOOL CHANGE ILLEGAL Z AXIS COMMAND	A Z-axis move command was performed in the same block for M06 command.
0251	TOOL CHANGE ILLEGAL T COMMAND	An unusable T code was specified in M06Txx.
0253	G05 CAN NOT BE COMMANDED	A binary operation was specified during advanced preview control mode.

Number	Message	Description
0300	ILLEGAL COMMAND IN SCALING	An illegal G code was specified during scaling. Modify the program. For the T system, one of the following functions is specified during scaling, this alarm is generated. - finishing cycle (G70 or G72) - outer surface rough-cutting cycle (G71 or G73) - end side rough-cutting cycle (G72 or G74) - closed loop cutting cycle (G73 or G75) - end side cutting-off cycle (G74 or G76) - outer surface or inner surface cutting-off cycle (G75 or G77) - multiple repetitive threading cycle (G76 or G78) - face drill cycle (G83 or G83) - face tap cycle (G84 or G84) - face boring cycle (G85 or G85) - side drill cycle (G87 or G87) - side tap cycle (G88 or G88) - side boring cycle (G89 or G89) - outer surface turning cycle or inner surface boring cycle (G77 or G20) - threading cycle (G78 or G21) - end side turning cycle (G79 or G24) (Specify G codes for systems B and C in that order.)
0301	RESETTING OF REFERENCE RETURN IS INHIBITED	Although bit 0 (IDGx) of parameter No. 1012 was set to 1 to inhibit the reference position from being set again for a return to the reference position without a dog, an attempt was made to perform a manual return to the reference position.
0302	SETTING THE REFERENCE POSITION WITHOUT DOG IS NOT PERFORMED	The reference position could not be set for a return to the reference position without a dog. Possible causes are: - The axis was not moved in the direction of a return to the reference position for jog feeding. - The axis was moved in the direction opposite to the direction of a manual return to the reference position. - Since the one-rotation signal sent from the position detector is not detected, the grid for manual reference position return is not established. (Bit 6 of diagnostic data No. 0201 must be 1.)
0303	REFERENCE POSITION RETURN IS NOT PERFORMED	When the setting of a reference position at any position was possible in Cs contour control (bit 0 (CRF) of parameter No. 3700 = 1), a G00 command was issued for the Cs contour axis without a return to the reference position after the serial spindle was switched to Cs contour control mode. Perform a reference position return with a G28 command before issuing a G00 command.
0304	G28 IS COMMANDED WITHOUT ZERO RETURN	Although a reference position was not set, an automatic return to the reference position (G28) was commanded.
0305	INTERMEDIATE POSITION IS NOT ASSIGNED	Although a G28 (automatic return to the reference position), G30 (return to the second, third, or fourth reference position), or G30/1 (return to the floating reference position) command was not issued after power-up, G29 (return from the reference position) was commanded.
0306	MISMATCH AXIS WITH CNR/CHF	The correspondence between the moving axis and the I, J, or K command is incorrect in a block in which chamfering is specified.
0307	CAN NOT START REFERENCE RETURN WITH MECHANICAL STOPPER SETTING	An attempt was made to set a butt-type reference position for an axis for which to use the function to set a reference position without a dog.

Number	Message	Description
0308	G72.1 NESTING ERROR	G72.1 was specified again during G72.1 rotation copying.
0309	G72.2 NESTING ERROR	G72.2 was specified again during G72.2 parallel copying.
0310	FILE NOT FOUND	The specified file could not be found during a subprogram or macro call.
0311	CALLED BY FILE NAME FORMAT ERROR	An invalid format was specified to call a subprogram or macro using a file name.
0312	ILLEGAL COMMAND IN DIRECT DRAWING DIMENSIONS PROGRAMMING	Direct input of drawing dimensions was commanded in an invalid format. An attempt was made to specify an invalid G code during direct input of drawing dimensions. Two or more blocks not to be moved exist in consecutive commands that specify direct input of drawing dimensions. Although non-use of commas (,) (bit 4 of parameter No. 3405 = 1) was specified for direct input of drawing dimensions, a comma was specified.
0313	ILLEGAL LEAD COMMAND	The variable-lead threading increment specified in address K exceeds the specified maximum value in variable-lead threading. Or, a negative lead value was specified.
0314	ILLEGAL SETTING OF POLYGONAL AXIS	An axis was specified invalidly in polygon turning. For polygon turning: A tool rotation axis is not specified. (Parameter No. 7610) For polygon turning between spindles: Valid spindles are not specified. (Parameters Nos. 7640 to 7643) - A spindle other than the serial spindle. - A spindle is not connected. For concurrent use of polygon turning and polygon turning with two spindles: - In the polygon turning mode, the value of parameter No. 7605 (selecting the type of polygon turning) was changed. - An attempt is made to use a spindle used for polygon turning also for polygon turning with two spindles.
0315	ILLEGAL NOSE ANGLE COMMAND IS IN THE THREAD CUTTING CYCLE	An invalid tool tip angle is specified in a multiple repetitive canned threading cycle (G76).
0316	ILLEGAL CUTTING AMOUNT IS IN THE THREAD CUTTING CYCLE	A minimum depth of cut higher than the thread height is specified in a multiple repetitive canned threading cycle (G76).
0317	ILLEGAL THREAD COMMAND IS IN THE THREAD CUTTING CYCLE	A zero or a negative value is specified in a multiple repetitive canned threading cycle (G76) as the thread height or the depth of cut.
0318	ILLEGAL RELIEF AMOUNT IS IN THE DRILLING CYCLE	Although an escape directions is set in a multiple repetitive canned cutting-off cycle (G74 or G75), a negative value is specified for Δd.
0319	THE END POINT COMMAND IS ILLEGAL IN THE DRILLING CYCLE	Although the Δi or Δk travel distance is set to 0 in a multiple repetitive canned cutting-off cycle (G74 or G75), a value other than 0 us specified for a U or W.
0320	ILLEGAL MOVEMENT AMOUNT/CUTTING AMOUNT IS IN THE DRILLING CYCLE	A negative value is specified in a multiple repetitive canned cutting-off cycle (G74 or G75) as Δi or Δk (travel distance/the depth of cut).
0321	ILLEGAL REPEATED TIME IS IN THE PATTERN REPEATING CYCLE	A zero or a negative value is specified in a multiple repetitive canned closed loop cycle (G73) as a repeated time.
0322	FINISHING SHAPE WHICH OVER OF STARTING POINT	An invalid shape which is over the cycle starting point is specified in a shape program for a multiple repetitive canned rough-cutting cycle (G71 or G72).

Number	Message	Description
0323	THE FIRST BLOCK OF SHAPE PROGRAM IS A COMMAND OF TYPE II	Type II is specified in the first block of the shape program specified by P in a multiple repetitive canned rough-cutting cycle (G71 or G72). Z (W) command is for G71. X (U) command is for G72.
0324	THE INTERRUPTION TYPE MACRO WAS DONE IN THE MULTIPLE REPETIVE CYCLES	An interruption type macro was issued during the multiple repetitive canned cycle (G70, G71, G72, or G73).
0325	UNAVAILABLE COMMAND IS IN SHAPE PROGRAM	An usable command was issued in a shape program for a multiple repetitive canned cycle (G70, G71, G72, or G73).
0326	LAST BLOCK OF SHAPE PROGRAM IS A DIRECT DRAWING DIMENSIONS	In a shape program in the multiple repetitive canned cycle (G70, G71, G72, or G73), a command for direct input of drawing dimensions in the last block is terminated in the middle.
0327	MODAL THAT MULTIPLE REPETIVE CYCLES CANNOT BE DONE	A multiple repetitive canned cycle (G70, G71, G72, or G73) was commanded in a modal state in which a multiple repetitive canned cycle could not be commanded.
0328	ILLEGAL WORK POSITION IS IN THE TOOL NOSE RADIUS COMPENSATION	The specification for the blank side for a tool nose radius compensation (G41 or G42) is incorrect in a multiple repetitive canned cycle (G71 or G72).
0329	THE FINISHING SHAPE IS NOT A MONOTONOUS CHANGE(SECOND AXES)	In a shape program for the multiple repetitive canned rough-cutting cycle (G71 or G72), the command of the second plane axis was not a monotonous increase or decrease.
0330	ILLEGAL AXIS COMMAND IS IN THE TURNING CANNED CYCLE	An axis other than the plane is specified n a canned cycle(G90, G92, or G94).
0331	ILLEGAL AXIS NUMBER IN AX[]	An illegal value is specified for an AX[] axis number.
0332	ILLEGAL AXIS ADDRESS IN AXNUM[]	An illegal value is specified for an AXNUM[] axis address.
0333	TOO MANY SPINDLE COMMANDS	Multiple spindle commands could be found in the same block in using an expansion spindle name. Only one spindle could be commanded in the same block.
0334	INPUT VALUE OUT OF EFFECTIVE RANGE	An offset data which was out of the effective range was specified. (malfunction prevention function)
0335	PLURAL M CODE	Multiple M codes are commanded simultaneously in a block for a wait function with peripheral devices by an M code.
0336	TOOL COMPENSATION COMMANDED MORE TWO AXES	For a tool length compensation C, an attempt was made to command the offset to other axes without canceling the offset. Or, for a tool length compensation C, multiple axes are specified in G43 or G44 block.
0337	EXCESS MAXIMUM INCREMENTAL VALUE	The command value exceeded the maximum amount of incremental. (malfunction prevention function)
0340	ILLEGAL RESTART(NANO SMOOTHING)	With manual absolute turned on, an attempt was made to restart the operation in nano smoothing mode after performing the manual interaction.
0341	TOO MANY COMMAND BLOCK (NANO SMOOTHING)	There are more blocks than can be commanded consecutively in nano smoothing mode.
0342	CUSTOM MACRO INTERRUPT ENABLE IN NANO SMOOTHING	A custom macro interrupt was enabled in nano smoothing mode. Or, nano smoothing mode was commanded with a custom macro interrupt enabled.
0343	ILLEGAL COMMAND IN NANO SMOOTHING	G43, G44, or G49 was commanded during a nano smoothing.
0344	CANNOT CONTINUE NANO SMOOTHING	An illegal command or operation by which a nano smoothing could not be continued was performed.
0345	TOOL CHANGE ILLEGAL Z AXIS POS	A tool change position on the Z-axis is incorrect.

Number	Message	Description
0346	TOOL CHANGE ILLEGAL TOOL NUM	A tool change position is not set.
0347	TOOL CHANGE ILLEGAL COMMAND IN SAME BLK.	Tool changing is commanded twice or more in the same block.
0348	TOOL CHANGE Z AXIS POS NOT ESTABLISHED	A tool change spindle on the Z-axis is not set.
0349	TOOL CHANGE SPINDLE NOT STOP	A tool change spindle stop is not stopped.
0350	PARAMETER OF THE INDEX OF THE SYNCHRONOUS CONTROL AXIS SET ERROR.	An illegal synchronization control axis number (parameter No. 8180) is set.
0351	BECAUSE THE AXIS IS MOVING, THE SYNC CONTROL IS CAN'T BE USED.	While the axis being subject to synchronization control was moving, an attempt was made to start or cancel the synchronization control by a synchronization control axis selection signal.
0352	SYNCHRONOUS CONTROL AXIS COMPOSITION ERROR.	This error occurred when: 1) An attempt was made to perform synchronization control for the axis during a synchronization, composition, or superposition. 2) An attempt was made to synchronize a further great-grandchild for a parent-child-grandchild relation. 3) An attempt was made to operate synchronization control although a parent-child-grandchild relation was not set.
0353	THE INSTRUCTION WAS DONE FOR THE AXIS WHICH WAS NOT ABLE TO MOVE.	This error occurred when: - For synchronization 1) A move command was issued to the axis for which bit 7 (NUMx) of parameter No. 8163 is set to 1. 2) A move command was issued to the slave axis. - For composition 1) A move command was issued to the axis for which bit 7 (NUMx) of parameter No. 8163 is set to 1. 2) A move command was issued to the axis for which bit 7 (MUMx) of parameter No. 8162 is set to 1.
0354	THE G28 WAS INSTRUCTED IN WITH THE REF POS NOT FIXED IN SYNC MODE	This error occurred when G28 was specified to the master axis being parking during synchronization control, but an axis reference position is not set for the slave axis.
0355	PARAMETER OF THE INDEX OF THE COMPOSITE CONTROL AXIS SET ERROR.	An illegal composite control axis number (parameter No. 8183) is specified.
0356	BECAUSE THE AXIS IS MOVING, THE COMP CONTROL IS CAN'T BE USED.	While the axis being subject to composite control was moving, an attempt was made to start or cancel the composite control by a composite control axis selection signal.
0357	COMPOSITE CONTROL AXIS COMPOSITION ERROR.	This error occurred when an attempt was made to perform composite control for the axis during a synchronization, composition, or superposition.
0359	THE G28 WAS INSTRUCTED IN WITH THE REF POS NOT FIXED IN COMP MODE	This error occurred when G28 was specified to the composite axis during composite control, but a reference position is not set to the other part of the composition.
0360	PARAMETER OF THE INDEX OF THE SUPERPOS CONTROL AXIS SET ERROR.	An illegal superposition control axis number (parameter No. 8186) is specified.
0361	BECAUSE THE AXIS IS MOVING, THE SUPERPOS CONTROL IS CAN'T BE USED.	While the axis being subject to superposition control was moving, an attempt was made to start or cancel the superposition control by a superposition control axis selection signal.

Number	Message	Description
0362	SUPERPOSITION CONTROL AXIS COMPOSITION ERROR.	This error occurred when: 1) An attempt was made to perform superposition control for the axis during a synchronization, composition, or superposition. 2) An attempt was made to synchronize a further great-grandchild for a parent-child-grandchild relation.
0363	THE G28 WAS INSTRUCTED IN TO THE SUPERPOS CONTROL SLAVE AXIS.	This error occurred when G28 was specified to the superposition control slave axis during superposition control.
0364	THE G53 WAS INSTRUCTED IN TO THE SUPERPOS CONTROL SLAVE AXIS.	This error occurred when G53 was specified to the slave axis being moved during superposition control.
0365	TOO MANY MAXIMUM SV/SP AXIS NUMBER PER PATH	The maximum control axis number or maximum control spindle number which could be used within a path was exceeded.
0366	IMPROPER G-CODE IN TURRET METHOD	When the turret change tools method was selected (bit 3 (TCT) of parameter No. 5040 = 0), G43, G43.1, G43.4, G43.5, or G43.7 was commanded.
0367	3-D CONV. WAS COMMANDED IN SYNC MODE AS THE PARAMETER PKUx(NO.8162#2) IS 0.	A 3-dimensional coordinate conversion was commanded during synchronization control when the bit 2 (PKUx) of parameter No. 8162 was 0.
0368	OFFSET REMAIN AT OFFSET COMMAND	- When the ATC change tools method was selected (bit 3 (TCT) of parameter No. 5040 = 1) during G43, G43.1, G43.4, or G43.5 mode, G43.7 was commanded. Or, G43, G43.1, G43.4, or G43.5 was commanded during G43.7 mode. - After the setting of bit 3 (TCT) of parameter No. 5040 was changed in a state in which tool offset remained, the tool offset command was specified.
0369	G31 FORMAT ERROR	- No axis is specified or tow or more axes are specified in the torque limit switch instruction (G31P98/P99). - The specified torque Q value in the torque limit switch instruction is out of range. The torque Q range is 1 to 254. - The high-speed continuous skip option is not present.
0370	G31P/G04Q ERROR	1) The specified address P value for G31 is out of range. The address P range is 1 to 4 in a multistage skip function. 2) The specified address Q value for G04 is out of range. The address Q range is 1 to 4 in a multistage skip function. 3) P1-4 for G31, or Q1-4 for G04 was commanded without a multistage skip function option. 4) <T series > The specified value of address P of G72 or G74 falls outside the range. Address P ranges from 1 to 4 in the multistage skip function. P1-4 was specified in G72 or G74 even though the multistage skip function option is not present.
0371	ILLEGAL FORMAT IN G10 OR L50	In a command format for a programmable parameter input, an attempt was made to change the parameter for an encryption (No. 3220), key (No. 3221), or protection range (No.3222 or No.3223) as a "the encryption function for the key and program." Modify the program.
0373	ILLEGAL HIGH-SPEED SKIP SIGNAL	In the skip commands (G31, G31P1 to G31P4) and dwell commands (G04, G04Q1 to G04Q4), the same high-speed signal is selected in different paths.

Number	Message	Description
0374	ILLEGAL REGISTRATION OF TOOL MANAGER(G10)	G10L75 or G10L76 data was registered during the following data registration: - From the PMC window. - From the FOCAS2. - By G10L75 or G10L76 in another system. Command G10L75 or G10L76 again after the above operation is completed.
0376	SERIAL DCL: ILLEGAL PARAMETER	1. When bit 1 of parameter No. 1815 is set to “1”, parameter No.2002#3 is set to “0” 2. The absolute-position detection function is enabled. (Bit 5 of parameter No. 1815 is set to “1”.)
0387	ILLEGAL RTM DI/DO VAR	There is no DI/DO variable that has a specified signal address (alphabet, number).
0389	ILLEGAL RTM SIGNAL BIT	Bits other than bits 0 to 7 cannot be specified with a DI/DO signal.
0390	ILLEGAL MACRO VAR	A macro variable which was not supported by the real time custom macro function was used.
0391	RTM BRANCH OVER	The number of branches supported with real time custom macros was exceeded.
0392	TOO MANY SENTENCE CONTROL	Many reserved words (ZONCE, ZEDGE, ZWHILE, ZDO, ZEND, G65, M99) for RTM control were used in a real time macro command.
0393	NO SENTENCE CONTROL	In a real time macro command, there is no data to be assigned.
0394	ILLEGAL SENTENCE CONTROL	The matching of reserved words (ZONCE, ZEDGE, ZWHILE, ZDO, ZEND, G65, M99) for RTM control is incorrect.
0395	ILLEGAL NC WORD CONTROL	Control code G65 or M99 for calling a subprogram or returning from a subprogram is not coded correctly.
0396	ILLEGAL RTM SENTENCE CONTROL	In other than a real time macro command, a reserved word (ZONCE, ZEDGE, ZWHILE, ZDO, or ZEND) for RTM control is used.
0397	RTM BUFFER OVER	There is no buffer available for real time macro commands. Too many blocks read in advance are buffered as triggers used by real time macro commands.
0398	'ID OVER IN BUFFER	In blocks read in advance, there are too many real time macro commands with the same ID.
0399	'ID EXECUTION IN SAME TIME	An attempt was made to execute real time macro commands with the same ID by using the same NC statement as a trigger.
0400	ONESHOT CMDOVER	Too many one-shot real time macro commands are specified.
0401	EXEC CMD NUM OVER IN SAME TIME	The number of real time macro commands that can be executed simultaneously was exceeded
0402	ILLEGAL TOKEN FOR RTM	A token, variable, or function that is not supported by the real time custom macro function was detected.
0403	ACCESS TO RTM PROTECT VAR	An attempt was made to access a protected variable.
0404	RTM ERROR	An error related to a real time macro command occurred.
0406	CODE AREA SHORTAGE	The storage size of the real time macro area is insufficient.
0407	DOULE SLASH IN RTM MODE	In the compile mode, an attempt was made to set the compile mode again.
0408	G90 IS NOT PERMITTED	The absolute command cannot be specified.
0409	ILLEGAL AXIS NO	An invalid axis number is specified.
0410	MIDDLE POINT IS NOT ZERO	An intermediate point other than 0 is specified with G28.
0411	SIMULTANEOUSLY AXES OVER	The maximum number of axes that can be controlled simultaneously was exceeded.
0412	ILLEGAL G CODE	An unusable G code was used.
0413	ILLEGAL ADDRESS	An unusable address was used.

Number	Message	Description
0414	ILLEGAL PMC AXIS NO.	An invalid PMC axis number is specified.
0415	GROUP IS IN USE	The group to which the specified axis belongs is already in used.
0416	UNABLE TO USE THE AXIS	The specified axis cannot be used.
0417	AXIS IS UNABLE TO MOVE	The specified axis is placed in the inoperative state.
0418	ILLEGAL FEED SETTING	An incorrect feedrate is set.
0419	ILLEGAL DISTANCE SETTING	A travel distance beyond the specifiable range is specified.
0420	CONSTANT NUMBER P	A subprogram is specified not by using a constant.
0421	ILLEGAL ARGUMENT G54	With G65, an invalid argument, L, is used.
0422	ILLEGAL ARGUMENT G54	With G65, an invalid argument is used.
0423	NO PMC AXIS CONTROL OPTION	The option for PMC axis control is missing.
0424	MULTIPLE AXES IN ONE GROUP	Multiple axes are using one group.
0425	ONE AXIS USE MULTIPLE GROU	One axis is using multiple groups.
0429	ILLEGAL COMMAND IN G10.6	When retract was started in a threading block, a retract command had been issued for the long axis direction of threading.
0430	TOOL LIFE PAIRS ZERO	Tool life management group number parameter No.6813 is 0.
0431	ILLEGAL T/R DATA OF TOOL LIFE	The arbitrary group number (T) or remaining amount setting (R) is invalid.
0432	UNAVAILABLE POSTURE IN TPC	A command specifying an impossible tool posture was issued in tool posture control. Review the machine configuration and command. In circular interpolation or helical interpolation under tool posture control, a command to reverse the tool posture with respect to the interpolation plane was specified. Review the machine configuration and command.
0436	ILLEGAL PARAMETER IN WSC	An incorrect parameter was specified in compensation of workpiece placement error. - he basic three axes are not specified in parameter No.1022.
0437	ILLEGAL COMMAND IN WSC	An invalid command related to compensation of workpiece placement error was specified. - An illegal G code was specified in the workpiece placement error compensation mode. - There is an error in the modal setting used when the compensation of workpiece placement error is started. - G54.4 was not specified solely. - There is not the P command in the block including the G54.4 command. Alternatively, the value following P is out of the range. - Compensation of workpiece placement error was specified redundantly.

Number	Message	Description
0438	ILLEGAL PARAMETER IN TOOL DIRC CMP	If, on a 5-axis machine, either of the two cases below applies, a parameter is illegal. <1> The setting is such that tool direction compensation is performed if workpiece setting error compensation is performed (bit 0 (RCM) of parameter No. 11200 = 1). <2> Tool center point retention type tool axis direction control (G53.6) is performed. - Acc./Dec. before interpolation is disabled. Set parameter No. 1660. - Acc./Dec. before rapid traverse interpolation is disabled. Set bit 1 of parameter No. 1401, bit 5 of parameter No. 1950, and parameter No. 1671. - The parameters (No.19680 to No.19714) for configuring the machine are incorrect. - The axis set by parameters Nos. 19681 and 19686 is not a rotation axis. - The basic three axes are not set in parameter No. 1022. - In tool length compensation during workpiece setting error compensation, bit 6 (TOS) of parameter No. 5006 is 0 and bit 2 (TOP) of parameter No. 11400 is 0. Set either parameter to 1.
0439	ILLEGAL COMMAND IN TOOL DIRC CMP	When compensation of workpiece placement error was performed in a 5-axis cutting machine (compensation in the tool direction (bit 0 (RCM) of parameter No. 11200 is 0)), an illegal command was issued. - An unspecifiable G code was specified. - There is an error in the modal setting used during startup. - An axis not related to 5-axis machining was specified. - The absolute coordinates of a rotation axis could not be obtained in the startup block of compensation of workpiece placement error or tool center point control.
0441	DUPLICATE PATH TABLE	The same Path Table numbers exist. Example) <AXIS_TABLE_1234_X1> and <TIME_TABLE_1234_X1> exist. <AXIS_TABLE_0001_M> and <TIME_TABLE_0001_M> exist.
0442	PATH TABLE COMMAND EXCES ERR	1. At the start of the Path Table Operation, the difference between the actual axis position and the start command at the Path Table exceeds the parameter No.11101. 2. At the start of the Path Table Operation, the difference between the actual spindle speed and the start command of spindle speed at the Path Table exceeds the parameter No.11102.
0443	PTRDY SIGNAL IS OFF	Even though the Path Table Ready signal PTRDY <Fn519.6> is "0", the Path Table Operation is started. Retry Path Table Operation after Path Table conversion.
0444	ILLEGAL PATH TABLE M-CODE	M/P/Q code for starting the Path Table operation is not correct.
0445	ILLEGAL AXIS OPERATION	The positioning command was issued in the speed control mode. Check the SV speed control mode signal.
0446	ILLEGAL COMMAND IN G96.1/G96.2/G96.3/G96.4	G96.1, G96.2, G96.3, and G96.4 are specified in the block that includes other commands. Modify the program.
0447	ILLEGAL SETTING DATA	The live tool axis is incorrectly set. Check the parameter for the spindle control function with servo motor.

Number	Message	Description
0451	ILLEGAL AUXILIARY FUNCTION TABLE COMMAND	When M code is output, the ladder of the PMC does not execute the completion processing of the previous M code.
0452	ILLEGAL PATH TABLE OPERATION	In Path Table Operation, the following problems occurred. - Skip command is not correct. - The connection of the Path Table is not correct. - Path Table Operation is not correct for other reason. etc. The detail alarm number is read by using the cnc_rdptexedistalm function. The detail alarm number can be read by using the C Language Executor or FOCAS2 cnc_ rdptexedistalm function. For details of cnc_rdptexedistalm function, refer to “CNC/PMC window library” in “C Language Executor Programming Manual (B-63943EN-3).
0455	ILLEGAL COMMAND IN GRINDING	In grinding canned cycles: 1) <M series> The signs of the I, J, and K commands do not match. 2) <M series/T series > The amount of travel of the grinding axis is not specified.
0456	ILLEGAL PARAMETER IN GRINDING	Parameters related to grinding canned cycles are incorrectly set. Probable causes are given below. 1) <M series/T series> The axis number of the grinding axis is incorrectly set (parameters Nos. 5176 to 5179). 2) <M series> The axis number of the dressing axis is incorrectly set (parameters Nos. 5180 to 5183). 3) <M series/T series> The axis numbers of the cut axis, grinding axis, and dressing axis (only for the M series) overlap.
0459	ALL PARALLEL AXES IN PARKING	All the axes specified during automatic operation are parking.
0460	ILLEGAL TORCH AXIS NUMBER	The axis number set in parameter No. 5490 (torch control axis) exceeds the number of control axes.
0461	ILLEGAL SETTING OF ROTATE AXIS FOR TORCH	The parameter setting (bit 0 of parameter No. 1006 = 1) of the rotation axis is not applied to the torch turning axis.
0492	3DCHK FIG. ILLEGAL: [Target name]	The figure data of [Target name] specified for built-in 3D interference check is invalid.
0493	3DCHK AXIS ILLEGAL: [Target name]	The move axis data of [Target name] specified for built-in 3D interference check is invalid.
0494	3DCHK FUNCTION INVALID	Built-in 3D interference check function is disabled by bit 0 (ICE) of parameter No. 10930.
0495	3DCHK TOO MANY FIGURE	The total number of shapes included in all interference check targets except the tool exceeds 23.
0496	ILLEGAL P,Q COMMAND IN G22.2	As for G22.2 command, parameter P or Q is out of range or not specified. Please correct G22.2 command.
0497	CANNOT MAKE TOOL FIGURE BY TOOL MANAGEMENT	According to bit 2 (ICT) of parameter No.10930, though the tool figure should be automatically made by using the tool management function, the option for the tool management function is not available.
0501	THE COMMANDED M-CODE CAN NOT BE EXECUTED	The M code specified in parameter No. 11631 to 11646 was specified in other than an execution macro, macro interrupt, macro call using a G or M code, or subprogram call using a T, S, or second auxiliary function code.
0502	ILLEGAL G-CODE	1) A G code unavailable in the inter-path flexible synchronous mode was specified. 2) A G code unavailable in the advanced superimposition state was specified.

Number	Message	Description
0503	ILLEGAL MODAL IN SUPERIMPOSED MODE	Modal state of a G code that cannot be superimposed.
0507	ILLEGAL PARAMETER(NO.7526)	The address of the R signal for the high-speed cycle machining operation information output function is invalid. 1) The specified R signal address is invalid. 2) The start address is not a multiple of 4 (0, 4, 8, ...). 3) A 36-byte area is not allocated.
0508	G code to need G90(PAC)	In parallel axis control, a G code requiring an absolute command (G90) in the block immediately before was specified.
0509	TOOL OFFSET COMMAND IS NOT AVAILABLE	- Tool offset (for the lathe system) was specified in the thread cutting block. - Tool offset was specified (lathe system) in the scaling mode, coordinate system rotation mode, or programmable mirror image mode.
0511	CS HI-SPEED SWITCHING FORMAT ERROR	The format of Cs contour control high speed switching is invalid.
0512	IMPOSSIBLE COMMAND FOR CS HI-SPEED SWITCHING	The following commands cannot be specified in Cs contour control high speed switching: - Move command not for high-speed cycle machining - Synchronous/composite control, superimposed control - Spindle command synchronous control - Simple spindle electronic gear box - Manual reference position return
0513	CS HI-SPEED SWITCHING SETTING ERROR	The setting for Cs contour control high speed switching is invalid. Possible causes are: - An M code value for Cs contour control high speed switching is used for multiple Cs contour control axes. - FIN is returned for the M code for high-speed switching of Cs contour control when the Cs contour control high speed switching completion signal CSMCx does not become 1. - The spindle software does not support the spindle control switching function for high-speed cycle machining.
0514	ILLEGAL COMMAND IN FLEXIBLE PATH AXIS ASSIGNMENT	1) An assignment command in flexible path axis assignment was issued for an axis yet to be removed. 2) The P, Q, R, I, J, K, or L value specified by G52.1, G52.2, or G52.3 is invalid. 3) The value of the parameter No. 11560 is duplicated. 4) An attempt was made to execute a removal command (G52.1) for an axis already removed. 5) An attempt was made to exchange axes having different settings of bit 1 (FAN) of parameter No. 11562. 6) An attempt was made to perform flexible path axis assignment without canceling the offset.
0515	ILLEGAL FORMAT IN SMOOTH TCP(G43.4L1)	An illegal command was specified in Rotation axes compensation (G43.4L1). • An illegal command was specified in a Rotation axes control start block. - An invalid value was specified with address "L". A value other than 0 and 1 was specified with address “L”. - G10.8 was specified at the same time.
0516	ILLEGAL PARAMETER IN SMOOTH TCP(G43.4L1)	A parameter related to in Rotation axes compensation (G43.4L1) is illegal.

Number	Message	Description
0517	SETTING ERROR AMOUNT IS OUT OF RANGE	An attempt was made to start workpiece setting error compensation when a rotation direction setting error was outside the range set in the corresponding parameter No. 11753 to 11758.
0520	ILLEGAL FORMAT IN G10.8L1	An illegal command was specified to change a tolerance of in Rotation axes compensation (G43.4L1). • A negative value was specified as a tolerance. - Specify positive values as addresses "α" and "β". • An invalid P value was specified. - Specify either 0 or 1 as address "P". • Address P is specified together with addresses "α" and "β". - Specify only either address "P" or "addresses "α" and "β". • An invalid address was specified. - In G10.8L1, only L, P, α, β, O, N, and M can be specified. • Another G code was specified at the same time. - Specify G10.8L1 alone. • G10.8 was specified in Rotation axes compensation mode (G43.4L1), but address "L" is not specified or the value of address "L" is not 1. - In Rotation axes compensation mode (G43.4L1), only G10.8L1 can be specified.
0521	ILLEGAL USAGE OF G10.8L1	Modal information used when specifying G10.8L1 contains an error. • The system is not in Rotation axes compensation mode. - G10.8L1 can be specified in Rotation axes compensation mode only. • The system is in Rotation axes compensation mode, but the command is not linear interpolation (G01). - G10.8L1 can be specified only during linear interpolation (G01).
0523	ILL. COMMAND IN HIGH-SPEED S-TCP	An illegal command was specified during Smooth control (G43.4P3). - An command G43.4 P3 was specified without option of High-speed Smooth TCP.
0525	SUB PROGRAM/MACRO CALLING	The mode was changed and the cycle start was executed while executing subprogram call or macro call. Reset and cycle start is executed again.
0527	ILLEGAL DATA IN PITCH ERROR	In the effective pitch error compensation points that is decided by considering the setting of the parameters, the difference between two consecutive data exceeds the range of -128 to +127. Please correct the pitch error compensation data or change the parameter.
0533	ILL. PARAM IN HIGH-SPEED S-TCP	A parameter related to Smooth control (G43.4P3) is illegal. - When setting the following, Smooth control (G43.4P3) was commanded. - the rotation axis is a hypothetical axis PRM.IA1,IA2(19696#0,1) - the inclination angle (PRM.19683,19688) - the reference angle RA,RB (PRM.19698,19699)

Number	Message	Description
0535	TL OFS CONVERSION SETTING ERROR	In the following cases, this alarm will occur when G44.1 is commanded. (1) The rotation axis is not set to the swivel head axis. (No. 19640#0) (2) The same rotation axis is set to both the tool nose rotation axis and the swivel head axis. (3) The axis direction of the tool nose rotation axis is not X-axis or Z-axis. (4) The axis direction of the swivel head axis is not Y-axis. (5) The number of axis for converted offset is short. (No. 1022)
0536	ILLEGAL COMMAND IN G44.1	An illegal command is specified in the tool offset conversion function. (1) An offset number is not commanded in G44.1 block. (2) Even if the imaginary tool nose direction is commanded in G44.1 block, the commanded value is not in range of 1 to 8. (3) The offset number is specified by D code that is different from the offset number specified by H code in G44.1 mode. (Machining center system only.) (4) Not allowed G code is specified at G44.1 block. (5) The rotation of the tool nose rotation axis or the swivel head axis is specified in G44.1 block. (6) The rotation of the tool nose rotation axis or the swivel head axis is specified in the block in which basic three axes movement are commanded. (7) In G44.1 mode, even if the rotation axis that is used in calculation of the tool offset conversion function is moved, G44.1 did not be commanded again before offset axis is moved. (8) G44.1 is commanded in the following mode. - Tool center point control. - Workpiece setting error compensation.
0601	ILLEGAL AXIS COMMAND FOR SERVO MOTOR SPINDLE	The move command was executed to the servo axes for spindle use of the spindle control with servo motor. Modify the program.
1001	AXIS CONTROL MODE ILLEGAL	Axis control mode is illegal.
1013	ILLEGAL POS. OF PROGRAM NO.	Address O or N is specified in an illegal location (e.g. after a macro statement).
1014	ILLEGAL FORMAT OF PROGRAM NO.	Address O or N is not followed by a number.
1016	EOB NOT FOUND	EOB (End of Block) code is missing at the end of a program input in the MDI mode.
1059	COMMAND IN BUFFERING MODE	The manual intervention compensation request signal MIGET became “1” when a advanced block was found during automatic operation. To input the manual intervention compensation during automatic operation, a sequence for manipulating the manual intervention compensation request signal MIGET is required in an M code instruction without buffering.
1077	PROGRAM IN USE	An attempt was made in the foreground to execute a program being edited in the background. The currently edited program cannot be executed, so end editing and restart program execution.
1079	PROGRAM FILE NOT FOUND	The program of the specified file No. is not registered in an external device. (external device subprogram call)

Number	Message	Description
1080	DUPLICATE DEVICE SUB PROGRAM CALL	Another external device subprogram call was made from a subprogram after the subprogram called by the external device subprogram call.
1081	EXT DEVICE SUB PROGRAM CALL MODE ERROR	The external device subprogram call is not possible in this mode.
1090	PROGRAM FORMAT ERROR	A lowercase alphabetic character is found in other than an NC program statement comment section, program name, or folder name.
1091	DUPLICATE SUB-CALL WORD	More than one subprogram call instruction was specified in the same block.
1092	DUPLICATE MACRO-CALL WORD	More than one macro call instruction was specified in the same block.
1093	DUPLICATE NC-WORD & M99	An address other than O, N, P or L was specified in the same block as M99 during the macro modal call state.
1095	TOO MANY TYPE-2 ARGUMENT	More than ten sets of I, J and K arguments were specified in the type–II arguments (A, B, C, I, J, K, I, J, K, ...) for custom macros.
1096	ILLEGAL VARIABLE NAME	An illegal variable name was specified. A code that cannot be specified as a variable name was specified. [#_OFSxx] does not match the tool offset memory option configuration.
1097	TOO LONG VARIABLE NAME	The specified variable name is too long.
1098	NO VARIABLE NAME	The specified variable name cannot be used as it is not registered.
1099	ILLLEGAL SUFFIX [ ]	A suffix was not specified to a variable name that required a suffix enclosed by [ ]. A suffix was specified to a variable name that did not require a suffix enclosed by [ ]. The value enclosed by the specified [ ] was out of range.
1100	CANCEL WITHOUT MODAL CALL	Call mode cancel (G67) was specified even though macro continuous–state call mode (G66) was not in effect.
1101	ILLEGAL CNC STATEMENT IRT.	An interrupt was made in a state where a custom macro interrupt containing a move instruction could not be executed.
1115	READ PROTECTED VARIABLE	An attempt was made in a custom macro to use on the right side of an expression a variable that can only be used on the left side of an expression.
1120	ILLEGAL ARGUMENT FORMAT	The specified argument in the argument function (ATAN, POW) is in error.
1124	MISSING DO STATEMENT	The DO instruction corresponding to the END instruction was missing in a custom macro.
1125	ILLEGAL EXPRESSION FORMAT	The description of the expression in a custom macro statement contains an error. A parameter program format error. The screen displayed to enter periodic maintenance data or item selection menu (machine) data does not match the data type.
1128	SEQUENCE NUMBER OUT OF RANGE	The jump destination sequence No. in a custom macro statement GOTO instruction was out of range (valid range: 1 to 99999999).
1131	MISSING OPEN BRACKET	The number of left brackets ([) is less than the number of right brackets (]) in a custom macro statement.
1132	MISSING CLOSE BRACKET	The number of right brackets (]) is less than the number of left brackets ([) in a custom macro statement.
1133	MISSING '='	An equal sign (=) is missing in the arithmetic calculation instruction in a custom macro statement.
1134	MISSING ','	A delimiter (,) is missing in a custom macro statement.

Number	Message	Description
1137	IF STATEMENT FORMAT ERROR	The format used in the IF statement in a custom macro is in error.
1138	WHILE STATEMENT FORMAT ERROR	The format used in the WHILE statement in a custom macro is in error.
1139	SETVN STATEMENT FORMAT ERROR	The format used in the SETVN statement in a custom macro is in error.
1141	ILLEGAL CHARACTER IN VAR. NAME	The SETVN statement in a custom macro contacts a character that cannot be used in a variable name.
1142	TOO LONG V-NAME (SETVN)	The variable name used in a SETVN statement in a custom macro exceeds 8 characters.
1143	BPRNT/DPRNT STATEMENT FORMAT ERROR	The format used in the BPRINT statement or DPRINT statement is in error.
1144	G10 FORMAT ERROR	The G10 L No. contains no relevant data input or corresponding option. Data setting address P or R is not specified. An address not relating to the data setting is specified. Which address to specify varies according to the L No. The sign, decimal point or range of the specified address are in error.
1145	G10.1 TIME OUT	The response to a G10.1 instruction was not received from the PMC within the specified time limit.
1146	G10.1 FORMAT ERROR	The G10.1 instruction format is in error.
1152	G31.9/G31.8 FORMAT ERROR	The format of the G31.9(continuous high-speed skip function) or G31.8(EGB skip function / skip function for flexible synchronous control) block is erroneous in the following cases: - The axis was not specified in the G31.9 or G31.8 block. - Multiple axes were specified in the G31.9 or G31.8 block. - The P code was specified in the G31.9 or G31.8 block. - G31.8 was commanded out of flexible synchronous control mode. - The Q was specified out of range in flexible synchronous control mode.
1153	CANNOT USE G31.9	G31.9 cannot be specified in this modal state. This alarm is also generated when G31.9 is specified when a group 07 G code (e.g. tool radius compensation) is not canceled.
1160	COMMAND DATA OVERFLOW	An overflow occurred in the position data within the CNC. This alarm is also generated if the target position of a command exceeds the maximum stroke as a result of calculation such as coordinate conversion, offset, or introduction of a manual intervention amount.
1180	ALL PARALLEL AXES IN PARKING	All of the axis specified for automatic operation are parked.
1196	ILLEGAL DRILLING AXIS SELECTED	An illegal axis was specified for drilling in a canned cycle for drilling. If the zero point of the drilling axis is not specified or parallel axes are specified in a block containing a G code in a canned cycle, simultaneously specify the parallel axes for the drilling axis.
1200	PULSCODER INVALID ZERO RETURN	The grid position could not be calculated during grid reference position return using the grid system as the one–revolution signal was not received before leaving the deceleration dog. This alarm is also generated when the tool does not reach a feedrate that exceeds the servo error amount preset to parameter No. 1841 before the deceleration limit switch is left (deceleration signal *DEC returns to “1”).

Number	Message	Description
1202	NO F COMMAND AT G93	F codes in the inverse time specification mode (G93) are not handled as modal, and must be specified in individual blocks.
1223	ILLEGAL SPINDLE SELECT	1) An attempt was made to execute an instruction that uses the spindle although the spindle to be controlled has not been set correctly. 2) Interpolation type rigid tapping was specified in a path in which the Cs contour control function is not enabled.
1282	ILLEGAL COMMAND IN 3-D OFFSET	An illegal G code was specified in the 3-dimensional tool compensation mode.
1283	ILLEGAL IJK IN 3-D OFFSET	When bit 0 (ONI) of parameter No. 6029 is set to 1, I, J, and K commands are specified without the decimal point in three–dimensional tool compensation mode.
1298	ILLEGAL INCH/METRIC CONVERSION	An error occurred during inch/metric switching.
1300	ILLEGAL ADDRESS	The axis No. address was specified even though the parameter is not an axis–type while loading parameters or pitch error compensation data from a tape or by entry of the G10 parameter. Axis No. cannot be specified in pitch error compensation data.
1301	MISSING ADDRESS	The axis No. was not specified even though the parameter is an axis–type while loading parameters or pitch error compensation data from a tape or by entry of the G10 parameter. Or, data No. address N, or setting data address P or R are not specified.
1302	ILLEGAL DATA NUMBER	A non–existent data No. was found while loading parameters or pitch error compensation data from a tape or by entry of the G10 parameter. An invalid address R value is specified in a pattern program for each machining purpose on the high–speed high–precision setting screen. This alarm is also generated when illegal word values are found.
1303	ILLEGAL AXIS NUMBER	An axis No. address exceeding the maximum number of controlled axes was found while loading parameters from a tape or by entry of the G10 parameter.
1304	TOO MANY DIGIT	Data with too many digits was found while loading parameters or pitch error compensation data from a tape.
1305	DATA OUT OF RANGE	Out–of–range data was found while loading parameters or pitch error compensation data from a tape. The values of the data setting addresses corresponding to L Nos. during data input by G10 was out of range. This alarm is also generated when NC programming words contain out–of–range values.
1306	MISSING AXIS NUMBER	A parameter which requires an axis to be specified was found without an axis No. (address A) while loading parameters from a tape.
1307	ILLEGAL USE OF MINUS SIGN	Data with an illegal sign was found while loading parameters or pitch error compensation data from a tape, or by entry of the G10 parameter. A sign was specified to an address that does not support the use of signs.
1308	MISSING DATA	An address not followed by a numeric value was found while loading parameters or pitch error compensation data from a tape.

Number	Message	Description
1329	ILLEGAL MACHINE GROUP NUMBER	An machine group No. address exceeding the maximum number of controlled machine groups was found while loading parameters from a tape or by entry of the G10 parameter.
1330	ILLEGAL SPINDLE NUMBER	An spindle No. address exceeding the maximum number of controlled spindles was found while loading parameters from a tape or by entry of the G10 parameter.
1331	ILLEGAL PATH NUMBER	An path No. address exceeding the maximum number of controlled path was found while loading parameters from a tape or by entry of the G10 parameter.
1332	DATA WRITE LOCK ERROR	Could not load data while loading parameters, pitch error compensation data and work coordinate data from tape.
1333	DATA WRITE ERROR	Could not write data while loading data from tape.
1360	PARAMETER OUT OF RANGE (TLAC)	Illegal parameter setting. (Set value is out of range.)
1361	PARAMTER SETTING ERROR 1 (TLAC)	Illegal parameter setting. (axis of rotation setting)
1362	PARAMETER SETTING ERROR 2 (TLAC)	Illegal parameter setting (tool axis setting)
1370	PARAMETER SETTING ERROR (DM3H-1)	Out–of–range data was set during setting of the three–dimensional handle feed parameter.
1371	PARAMETER SETTING ERROR (DM3H-2)	An illegal axis of rotation was set during setting of the three–dimensional handle feed parameter.
1372	PARAMETAR SETTING ERROR (DM3H-3)	An illegal master axis was set during setting of the three–dimensional handle feed parameter.
1373	PARAMETER SETTING ERROR (DM3H-4)	An illegal parallel axis or twin table was set during setting of the three–dimensional handle feed parameter.
1470	G40.1 –G42.1 PARAMETER MISS	A parameter setting related to normal direction control is illegal. The axis number of a normal direction controlled axis is set in parameter No. 5480, but that axis number is in the range of the number of controlled axes. The axis set as a normal direction controlled axis is not set as a rotation axis (bit 0 (ROTx) of parameter No. 1006) = 1 and No.1022=0).
1471	ILLEGAL COMMAND IN G40.1 –G42.1	An invalid G code was specified in the normal direction control mode.
1508	DUPLICATE M-CODE (INDEX TABLE REVERSING)	A function to which the same code as this M code is set exists. (index table indexing)
1509	DUPLICATE M-CODE (SPOS AXIS ORIENTATION)	A function to which the same code as this M code is set exists. (spindle positioning, orientation)
1510	DUPLICATE M-CODE (SPOS AXIS POSITIONING)	A function to which the same code as this M code is set exists. (spindle positioning, positioning)
1511	DUPLICATE M-CODE (SPOS AXIS RELEASE)	A function to which the same code as this M code is set exists. (spindle positioning, mode cancel)
1531	ILLEGAL USE OF DECIMAL POINT (F-CODE)	When the feedrate instruction contains valid data below the decimal point, the alarm is set and the F code contains valid data below the decimal point.
1532	ILLEGAL USE OF DECIMAL POINT (E-CODE)	When the feedrate instruction contains valid data below the decimal point, the alarm is set and the E code contains valid data below the decimal point.
1533	ADDRESS F UNDERFLOW (G95)	The feedrate for the hole drilling axis calculated from the F and S codes is too slow in the feed per single rotation mode (G95).
1534	ADDRESS F OVERFLOW (G95)	The feedrate for the hole drilling axis calculated from the F and S codes is too fast in the feed per single rotation mode (G95).

Number	Message	Description
1535	ADDRESS E UNDERFLOW (G95)	The feedrate for the hole drilling axis calculated from the E and S codes is too slow in the feed per single rotation mode (G95).
1536	ADDRESS E OVERFLOW (G95)	The feedrate for the hole drilling axis calculated from the E and S codes is too fast in the feed per single rotation mode (G95).
1537	ADDRESS F UNDERFLOW (OVERRIDE)	The speed obtained by applying override to the F instruction is too slow.
1538	ADDRESS F OVERFLOW (OVERRIDE)	The speed obtained by applying override to the F instruction is too fast.
1539	ADDRESS E UNDERFLOW (OVERRIDE)	The speed obtained by applying override to the E instruction is too slow.
1540	ADDRESS E OVERFLOW (OVERRIDE)	The speed obtained by applying override to the E instruction is too fast.
1541	S-CODE ZERO	“0” has been instructed as the S code.
1542	FEED ZERO (E-CODE)	“0” has been instructed as the feedrate (E code).
1543	ILLEGAL GEAR SETTING	The gear ratio between the spindle and position coder, or the set position coder number of pulses is illegal in the spindle positioning function.
1544	S-CODE OVER MAX	The S command exceeds the maximum spindle rotation number.
1548	ILLGAL AXIS MODE	The spindle positioning axis/Cs contour control axis was specified during switching of the controlled axis mode.
1561	ILLEGAL INDEXING ANGLE	The specified angle of rotation is not an integer multiple of the minimum indexing angle.
1564	INDEX TABLE AXIS – OTHER AXIS SAME TIME	The index table indexing axis and another axis have been specified in the same block.
1567	INDEX TABLE AXIS DUPLICATE AXIS COMMAND	Index table indexing was specified during axis movement or on an axis for which the index table indexing sequence was not completed.
1580	ENCODE ALARM (PSWD&KEY)	When an attempt was made to input a program, the specified password did not match the password on the tape and the password on tape was not equal to 0. When an attempt was made to output an encrypted tape, the password was not in the range 0 to 99999999. The password parameter is No. 2210.
1581	ENCODE ALARM (PARAMETER)	When an attempt was made to output an encrypted tape, the output code parameter was set to EIA. Set bit 1 (ISO) of parameter No. 0000 to “0”. An incorrect instruction was specified for program encryption or protection. This alarm is generated if an attempt is made to perform program editing, deletion, or range-specified output in the protected range in the lock state. Or, a program outside the protected range is specified in rage specification output in the unlock state. The protected range is defined from the program No. preset by parameter No. 3222 up to the program No. preset to parameter No. 3223. When both parameters are set to “0”, the protected range becomes O9000 to O9999.
1590	TH ERROR	A TH error was detected during reading from an input device. The read code that caused the TH error and how many statements it is from the block can be verified in the diagnostics screen.
1591	TV ERROR	An error was detected during the single–block TV error. The TV check can be suppressed by setting bit 0 (TVC) of parameter No. 0000 to “0”.

Number	Message	Description
1592	END OF RECORD	The EOR (End of Record) code is specified in the middle of a block. This alarm is also generated when the percentage at the end of the NC program is read. For the program restart function, this alarm is generated if a specified block is not found.
1593	EGB PARAMETER SETTING ERROR	Error in setting a parameter related to the EGB (1) The setting of SYN, bit 0 of parameter No. 2011, is not correct. (2) The slave axis specified with G81 is not set as a rotation axis. (bit 0 (ROT) of parameter No. 1006) (3) Number of pulses per rotation (Parameter No. 7772, 7773, 7782, or 7783 is not set.) (4) For a hobbing-machine-compatible command, parameter No. 7710 is not specified. (5) The slave axis specified with G81 for the spindle EGB is set as the slave axis for the simple spindle EGB. (6) No signal-based EGB synchronization ratio (parameters Nos. 7784 and 7785) has been set.
1594	EGB FORMAT ERROR	Error in the format of the block of an EGB command (1) T (number of teeth) is not specified in the G81 block. (2) In the G81 block, the data specified for one of T, L, P, and Q is out of its valid range. (3) n the G81 block, only one of P and Q is specified. (4) In the G81.5 block, there is no command for the master or slave axis. (5) In the G81.5 block, data out of the specified range is specified for the master or slave axis.
1595	ILL-COMMAND IN EGB MODE	During synchronization with the EGB, a command that must not be issued is issued. (1) Slave axis command using G27, G28, G29, G30,G30.1, G33, G53, etc. (2) Inch/metric conversion command using G20, G21, etc. (3) Synchronization start command using G81 or G81.5 when bit 3 (ECN) of parameter No. 7731 is 0 (4) For the slave axis of the spindle EGB, the Cs contour control mode is not selected.
1596	EGB OVERFLOW	An overflow occurred in the calculation of the synchronization coefficient.
1597	EGB AUTO PHASE FORMAT ERROR	Format error in the G80 or G81 block in EGB automatic phase synchronization (1) R is outside the permissible range. (2) In the spindle EGB, reference position return is not performed for the master axis before G81R2 is specified.
1598	EGB AUTO PHASE PARAMETER SETTING ERROR	Error in the setting of a parameter related to EGB automatic phase synchronization (1) The acceleration/deceleration parameter is not correct. (2) The automatic phase synchronization parameter is not correct.
1618	ILLEGAL P-DATA(WHEEL WEAR COMPENSATION)	There is an error in P-data in compensation selection of grinding wheel wear compensation. Alternatively, the P command is not present.

Number	Message	Description
1619	ILLEGAL AXIS(WHEEL WEAR COMPENSATION)	The compensation axis was switched in the grinding wheel wear compensation mode or compensation vector hold mode. Alternatively, parameters Nos. 5071 and 5072, which determine the axis to be subjected to grinding wheel wear compensation, are incorrectly set.
1805	ILLEGAL COMMAND	[I/O Device] An attempt was made to specify an illegal command during I/O processing on an I/O device. [G30 Zero Return] The P address Nos. for instructing No. 2 to No. 4 zero return are each out of the range 2 to 4. [Single Rotation Dwell] The specified spindle rotation is “0” when single rotation dwell is specified. [3-dimensional tool compensation] A G code that cannot be specified was specified in the 3-dimensional tool compensation mode. Scaling instruction G51, skip cutting G31 and automatic tool length measurement G37 were specified.
1806	DEVICE TYPE MISS MATCH	An operation not possible on the I/O device that is currently selected in the setting was specified. This alarm is also generated when file rewind is instructed even though the I/O device is not a FANUC Cassette.
1807	PARAMETER SETTING ERROR	An I/O interface option that has not yet been added on was specified. The external I/O device and baud rate, stop bit and protocol selection settings are erroneous.
1808	DEVICE DOUBLE OPENED	An attempt was made to open a device that is being accessed.
1809	ILLEGAL COMMAND IN G41/G42	Specified direction tool length compensation parameters are incorrect. A move instruction for a axis of rotation was specified in the specified direction tool length compensation mode.
1820	ILLEGAL DI SIGNAL STATE	1. An each axis workpiece coordinate system preset signal was turned “1” in the state in which all axes on the path including the axis on which to perform preset with the each axis workpiece coordinate system were not stopped or in which a command was in execution. 2. When an M code for performing preset with an each axis workpiece coordinate system preset signal was specified, the each axis workpiece coordinate system preset signal was not turned “1”. 3. The auxiliary function lock is enabled. 4. When bit 6 (PGS) of parameter No. 3001 was set to 0 (M, S, T, and B codes are not output in the high speed program check mode), an M code for turning “1” an each axis workpiece coordinate system preset signal in the high speed program check mode was specified.
1823	FRAMING ERROR(1)	The stop bit of the character received from the I/O device connected to RS232C interface 1 was not detected.
1830	DR OFF(2)	The data set ready input signal DR of the I/O device connected to RS232C interface 2 turned OFF.
1832	OVERRUN ERROR(2)	The next character was received from the I/O device connected to RS232C interface 2 before it could read a previously received character.

Number	Message	Description
1833	FRAMING ERROR(2)	The stop bit of the character received from the I/O device connected to RS232C interface 2 was not detected.
1834	BUFFER OVERFLOW(2)	The NC received more than 10 characters of data from the I/O device connected to RS232C interface 2 even though the NC sent a stop code (DC3) during data reception.
1889	ILLEGAL COMMAND IN G54.3	An illegal command was issued in G54.3 block. (1) An attempt was made to command G54.3 in a mode in which it cannot be accepted. (2) The command was not issued in a single block.
1892	LLEGAL PARAMETER IN G43.3	A parameter related to nutating rotary head tool length compensation is incorrect.
1893	ILLEGAL PARAMETER IN G44.9	A parameter related to spindle unit compensation is incorrect.
1898	ILLEGAL PARAMETER IN G54.2	Illegal parameters Nos. 6068 to 6076 were specified for fixture offset.
1912	V-DEVICE DRIVER ERROR (OPEN)	An error occurred during device driver control.
1919	FATAL ERROR(USB MEMORY)	A fatal error occurred in the USB file system. To restore the file system, turn the power off.
1924	UNEXPECTED ERROR(USB MEMORY)	An unexpected error occurred in the USB file system.
1925	ILLEGAL PATH/FILE(USB MEMORY)	An invalid path or file name was specified.
1926	ACCESS DENIED(USB MEMORY)	The USB memory could not be accessed.
1927	DEVICE IN FORMATTING(USB MEMORY)	The USB memory is being formatted.
1928	DEVICE NOT FOUND(USB MEMORY)	No USB memory is inserted. Check the connection.
1932	DEVICE IS FULL(USB MEMORY)	The capacity of the USB memory is insufficient.
1937	RECOGNITION ERROR(USB MEMORY)	The format of the USB memory is invalid. Format the USB memory in FAT or FAT32 format. If the alarm is still issued, replace the USB memory.
1938	END OF FILE FOUND(USB MEMORY)	The end of file was detected before EOR(%) was read. The file may be damaged.
1939	UNDEFINED ERROR(USB MEMORY)	An undefined error occurred.
1951	DEVICE IS BUSY(USB MEMORY)	The USB memory is busy.
1952	TOO MANY FILES(USB MEMORY)	The maximum number of files that can be opened concurrently is exceeded.
1953	REMOVED IN ACCESSING(USB MEMORY)	The USB memory was removed while being accessed.
1954	PATH/FILE EXIST(USB MEMORY)	The specified path or file already exists.
1955	PATH/FILE NOT FOUND(USB MEMORY)	The specified path or file is not found.
1956	DEVICE OVERCURRENT(USB MEMORY)	Overcurrent was detected in the USB memory. Replace the USB memory.
1957	PARITY ERROR(USB MEMORY)	A parity error occurred in the USB memory. Turn the power to the CNC off.
1960	ACCESS ERROR (MEMORY CARD)	Illegal memory card accessing This alarm is also generated during reading when reading is executed up to the end of the file without detection of the EOR code.
1961	NOT READY (MEMORY CARD)	The memory card is not ready.
1962	CARD FULL (MEMORY CARD)	The memory card has run out of space.
1963	CARD PROTECTED (MEMORY CARD)	The memory card is write–protected.
1964	NOT MOUNTED (MEMORY CARD)	The memory card could not be mounted.
1965	DIRECTORY FULL (MEMORY CARD)	The file could not be generated in the root directory for the memory card.
1966	FILE NOT FOUND (MEMORY CARD)	The specified file could not be found on the memory card.

Number	Message	Description
1967	FILE PROTECTED (MEMORY CARD)	The memory card is write–protected.
1968	ILLEGAL FILE NAME (MEMORY CARD)	Illegal memory card file name
1969	ILLEGAL FORMAT (MEMORY CARD)	Check the file name.
1970	ILLEGAL CARD (MEMORY CARD)	This memory card cannot be handled.
1971	ERASE ERROR (MEMORY CARD)	An error occurred during memory card erase.
1972	BATTERY LOW (MEMORY CARD)	The memory card battery is low.
1973	FILE ALREADY EXIST	A file having the same name already exists on the memory card.
1990	SPL:ILLEGAL AXIS COMMAND	The axis specified by the smooth interpolation (G5.1Q2) is illegal.
1993	SPL:CAN'T MAKE VECTOR	The end point and the 2 previous point are the same in generation of the 3–dimensional tool compensation vector by the end point for smooth interpolation.
1995	ILLEGAL PARAMETER IN G41.2/G42.2	The parameter settings (parameters Nos. 6080 to 6089) for determining the relationship between the axis of rotation and the rotation plane are incorrect.
1999	ILLEGAL PARAMETER IN G41.3	The parameter settings (parameters Nos. 6080 to 6089) for determining the relationship between the axis of rotation and the rotation plane are incorrect.
2002	NO KNOT COMMAND (NURBS)	Knot has not been specified, or a block not related to NURBS interpolation was specified in the NURBS interpolation mode.
2003	ILLEGAL AXIS COMMAND (NURBS)	An axis not specified as a control point was specified in the No. 1 block.
2004	ILLEGAL KNOT	There is an insufficient number of knot individual blocks.
2005	ILLEGAL CANCEL (NURBS)	The NURBS interpolation mode was turned OFF even though NURBS interpolation was not completed.
2006	ILLEGAL MODE (NURBS)	A mode that cannot be paired with the NURBS interpolation mode was specified.
2007	ILLEGAL MULTI-KNOT	Nested knots for each level can be specified for the start and end points.
2032	EMBEDDED ETHERNET/DATA SERVER ERROR	An error was returned in the built-in Ethernet/data server function. For details, see the error message screen of the built-in Ethernet or data server.
2038	WRONG COMMUNICATION COMBINATION	The combination of the hardware and the software about communication function is wrong. The detail information is indicated to diagnosis information 4400 and 4401.
2051	#200-#499ILLEGAL P-CODE MACRO COMMON INPUT(NO OPTION)	An attempt was made to enter a custom macro common variable not existing in the system.
2052	#500-#549P-CODE MACRO COMMON SELECT(CANNOT USE SETVN)	The variable name cannot be entered. The SETVN command cannot be used with the P-CODE macro common variables #500 to #549.
2053	THE NUMBER OF #30000 IS UNMATCH	An attempt was made to enter a P-CODE-only variable not existing in the system.
2054	THE NUMBER OF #40000 IS UNMATCH	An attempt was made to enter an extended P-CODE-only variable not existing in the system.
2060	ILLEGAL PARAMETER IN G43.4/G43.5	The parameter for the pivot tool length compensation is incorrect.

Number	Message	Description
2061	ILLEGAL COMMAND IN G43.4/G43.5	An illegal command was specified in tool center point control. - A rotation axis command was specified in tool center point control (type 2) mode. - With a table rotary type or mixed-type machine, a I, J, or K command was specified in the tool center point control (type 2) command (G43.5) block. - A command that does not move the tool center point (only a rotation axis is moved) was specified for the workpiece in the G02 mode. - G43.4 or G43.5 was specified in the tool center point control mode. - When the workpiece coordinate system is set as the programming coordinate system (bit 5 (WKP) of parameter No. 19696 is 1), G02 or G03 was specified while the rotation axis was not perpendicular to the plane.
2070	G02.1/ G03.1 FORMAT ERROR	- The format is invalid. - The specified arc exceeds the interpolation enable range.
4010	ILLEGAL REAL VALUE OF OBUF :	The real value for a output buffer is in error.
5006	TOO MANY WORD IN ONE BLOCK	The number of words in a block exceeds the maximum. The maximum is 26 words. However, this figure varies according to NC options. Divide the instruction word into two blocks.
5007	TOO LARGE DISTANCE	Due to compensation, point of intersection calculation, interpolation or similar reasons, a movement distance that exceeds the maximum permissible distance was specified. Check the programmed coordinates or compensation amounts.
5009	PARAMETER ZERO (DRY RUN)	The dry run feedrate parameter No. 1410 or maximum cutting feedrate parameter No. 1422 for each axis has been set to 0. The parameter for the maximum cutting feedrate for each axis is No. 1432 if acceleration/deceleration before interpolation is enabled and No. 1430 otherwise. Functions that cause acceleration/deceleration before interpolation include AI contour control, tool center point control, and workpiece setting error compensation.
5010	END OF RECORD	The EOR (End of Record) code is specified in the middle of a block. This alarm is also generated when the percentage at the end of the NC program is input.
5011	PARAMETER ZERO (CUT MAX)	The maximum cutting feedrate parameter No. 1430 has been set to 0. The parameter is No. 1432 if acceleration/deceleration before interpolation is enabled and No. 1430 otherwise. Functions that cause acceleration/deceleration before interpolation include AI contour control, tool center point control, and workpiece setting error compensation.
5014	TRACE DATA NOT FOUND	A transfer could not be made because of no trace data.
5015	NO ROTATION AXIS	No rotation axis was found in a handle feed in the tool axis direction or in the tool axis right angle direction.
5016	ILLEGAL COMBINATION OF M CODES	M codes which belonged to the same group were specified in a block. Alternatively, an M code which must be specified without other M codes in the block was specified in a block with other M codes.

Number	Message	Description
5018	POLYGON SPINDLE SPEED ERROR	In G51.2 mode, the speed of the spindle or polygon synchronous axis either exceeds the clamp value or is too small. The specified rotation speed ratio thus cannot be maintained. For polygon turning between spindles: More information as to why this alarm occurred is indicated in diagnosis data No. 471.
5020	PARAMETER OF RESTART ERROR	An invalid value is set in parameter No. 7310, which specifies the axis order in which the tool is moved along axes to the machining restart position in dry run. A value ranging from 1 to the number of controlled axes may be set in this parameter.
5043	TOO MANY G68 NESTING	3-dimensional coordinate conversion has been specified three or more times. To perform another coordinate conversion, perform cancellation, then specify the coordinate conversion.
5044	G68 FORMAT ERROR	Errors for 3-dimensional coordinate conversion command are: (1) No I, J, or K command was issued in 3-dimensional coordinate conversion command block. (without coordinate rotation option) (2) All of I, J, or K command were 0 in 3-dimensional coordinate conversion command block. (3) No rotation angle R was not commanded in 3-dimensional coordinate conversion command block.
5046	ILLEGAL PARAMETER (S-COMP)	The setting of a parameter related to straightness compensation contains an error. Possible causes include: - A non-existent axis number is set in a moving or compensation axis parameter. - More than 128 pitch error compensation points are set between the furthest points in the negative and position regions. - The straightness compensation point numbers do not have correct magnitude relationships. - No straightness compensation point is found between the furthest pitch error compensation point in the negative region and that in the positive region. - The compensation per compensation point is either too large or too small.
5050	ILL-COMMAND IN G81.1 MODE	During chopping/oscillation, a move command has been issued for the chopping/oscillation axis.
5058	G35/G36 FORMAT ERROR	A command for switching the major axis has been specified for circular threading. Alternatively, a command for setting the length of the major axis to 0 has been specified for circular threading.
5060	ILLEGAL PARAMETER IN G02.3/G03.3	The axis parameter setting to perform an exponential interpolation is in error. Parameter No. 5641: A liner axis number for performing an exponential interpolation Parameter No. 5642: A rotation axis number for performing an exponential interpolation The settable value is 1 to the number of control axes, but it must not be duplicated.

Number	Message	Description
5061	ILLEGAL FORMAT IN G02.3/G03.3	The exponential interpolation command (G02.3/G03.3) has a format error. The command range for address I or J is -89.0 to -1.0 or +1.0 to +89.0. No I or J is specified or out-of -range value is specified. No address R, or 0 is specified.
5062	ILLEGAL COMMAND IN G02.3/G03.3	The value specified in an exponential interpolation command (G02.3/03.3) is illegal. A value that does not allow exponential interpolation is specified. (For example, the value for In is 0 or negative.)
5064	DIFFERRENT AXIS UNIT	Circular interpolation has been specified on a plane consisting of axes having different increment systems.
5065	DIFFERRENT AXIS UNIT(PMC AXIS)	Axes having different increment systems have been specified in the same DI/DO group for PMC axis control. Modify the setting of parameter No. 8010.
5066	RESTART ILLEGAL SEQUENCE NUMBER	A sequence number from 7000 to 7999 was read during the search for the next number in a restart program for the back or restart function.
5068	FORMAT ERROR IN G31P90	No travel axis was specified. Two or more travel axes were specified.
5073	NO DECIMAL POINT	No decimal point has been specified for an address requiring a decimal point.
5074	ADDRESS DUPLICATION ERROR	The same address has been specified two or more times in a single block. Alternatively, two or more G codes in the same group have been specified in a single block.
5085	SMOOTH IPL ERROR 1	A block for specifying smooth interpolation contains a syntax error.
5110	IMPROPER G-CODE (AICC MODE)	An unspecifiable G code was specified in the AI contour control mode.
5115	ILLEGAL ORDER (NURBS)	There is an error in the specification of the rank.
5116	ILLEGAL KNOT VALUE (NURBS)	Monotone increasing of knots is not observed.
5117	ILLEGAL 1ST CONTROL POINT (NURBS)	The first control point is incorrect. Or, it does not provide a continuity from the previous block.
5118	ILLEGAL RESTART (NURBS)	After manual intervention with manual absolute mode set to on, NURBS interpolation was restarted.
5122	ILLEGAL COMMAND IN SPIRAL	A spiral interpolation or conical interpolation command has an error. Specifically, this error is caused by one of the following: 1) L = 0 is specified. 2) Q = 0 is specified. 3) R/, R/, C is specified. 4) Zero is specified as height increment. 5) Zero is specified as height difference. 6) Three or more axes are specified as the height axes. 7) A height increment is specified when there are two height axes. 8) Q is specified when radius difference = 0. 9) Q < 0 is specified when radius difference > 0. 10) Q > 0 is specified when radius difference < 0. 11) A height increment is specified when no height axis is specified.
5123	OVER TOLERANCE OF END POINT IN SPIRAL	The difference between a specified end point and the calculated end point exceeds the allowable range (parameter 3471).

Number	Message	Description
5124	CAN NOT COMMAND SPIRAL	A spiral interpolation or conical interpolation was specified in any of the following modes: 1) Scaling 2) Polar coordinate interpolation 3) In tool radius·tool nose radius compensation mode, the center is set as the end point.
5130	NC AND SUPERIMPOSE AXIS CONFLICT	In the PMC superposition axis control, the NC command and The PMC axis control command were conflicted. Modify the program and the ladder.
5131	NC COMMAND IS NOT COMPATIBLE	The PMC axis control and 3-dimensional coordinate conversion or a polar coordinate interpolation were specified simultaneously.
5132	CANNOT CHANGE SUPERIMPOSED AXIS	The superposition axis was selected for the axis for which the PMC superposition axis is being controlled.
5155	NOT RESTART PROGRAM BY G05	When learning control/preview repetitive control was enabled, an attempt was made to use feed hold or interlock to stop high-speed cycle machining/high-speed binary operation. Neither feed hold nor interlock can be used in such a case.
5195	DIRECTION CAN NOT BE JUDGED	Measurement is invalid in the tool compensation measurement value direct input B function. [For 1-contact input] 1. The recorded pulse direction is not constant. - The machine is at a stop in the offset write mode. - The servo power is off. - Pulse directions are diverse. 2. The tool is moving along the two axes (X-axis and Y-axis). [For the movement direction discrimination specification] 1. The recorded pulse direction is not constant. - The machine is at a stop in the offset write mode. - The servo power is off. - Pulse directions are diverse. 2. The tool is moving along the two axes (X-axis and Z-axis). 3. The direction indicated by the tool compensation write signal does not match the movement direction of the axis.
5196	ILLEGAL AXIS OPERATION	During HPCC or during the execution of a 5-axis-related function, an unavailable function was used.
5199	ILLEGAL FINE TORQUE SENSING PARAMETER	The setting of a fine torque sensing parameter is invalid. - An invalid controlled axis number is set as the target axis.
5219	CAN NOT RETURN	Manual intervention and return cannot be performed during execution of 3-dimensional coordinate system conversion, tilted working plane command, tool center point control, or work setting error compensation.
5220	REFERENCE POINT ADJUSTMENT MODE	In case of distance coded linear scale I/F, the reference point auto setting bit 2 of parameter No. 1819 is set to "1". Move the machine to reference position by manual operation and execute manual reference return.
5242	ILLEGAL AXIS NUMBER	A master axis number or a slave axis number was not set correctly when the flexible synchronization control mode was turned from off to on during automatic operation. In inter-path flexible synchronous control, this alarm is issued in either of the following cases. (The alarm is issued at the start of inter-path flexible synchronous control.) 1. The axis number of the master or slave axis is incorrect. 2. The master and slave axis settings make a loop.

Number	Message	Description
5243	DATA OUTRANGE	A gear ratio was not set correctly when the flexible synchronization control mode was turned from off to on during automatic operation.
5244	TOO MANY DI ON	• When an attempt was made to change the flexible synchronous control status, the select signal was not turned on or off after the execution of the M code. • An attempt was made to turn flexible synchronous control on or off without stopping the tool along all axes. (Except when automatic phase synchronization for flexible synchronous control is used) • Flexible synchronous control was turned off in any of the following function modes: - Tool center point control - Tilted working plane command - 3-dimensional cutter compensation - Workpiece setting error compensation
5245	OTHER AXIS ARE COMMANDED	- For a flexible synchronization control group for which a PMC axis was a master axis, an attempt was made to turn on the synchronous mode during time other than automatic operation. - An attempt was made to turn on a synchronization group for which an PMC axis was a master axis when there existed a flexible synchronization control group for which a non-PMC, normal axis was a master axis. - The master and slave axes as synchronous axes overlap the EGB dummy axis. - The master and slave axes as synchronous axes overlap the chopping axis. - The master and slave axes as synchronous axes overlap the axis related to angular axis control. - The master and slave axes as synchronous axes overlap the axis related to composite control. - The master and slave axes as synchronous axes overlap the axis related to superposition control. - The slave axis as a synchronous axis overlaps the axis related to synchronization control. - The reference position return mode is turned on (was turned on). - A servo alarm occurred in a path in inter-path flexible synchronous control. - An emergency stop was applied in another path in inter-path flexible synchronous control. - When an attempt was made to execute flexible synchronization between different paths during automatic operation, the inter-path flexible synchronous mode was not enabled.
5255	G12.4/G13.4 FORMAT ERROR	The specified P, I, and K are incorrect or I is less than K.
5256	G12.4/G13.4 EXECUTION ERROR	1) In continuous circle motion-based groove cutting mode, a command other than G01, G02, G03, G04, G90, G91, and auxiliary functions is specified. 2) In a mode that cannot be used, the continuous circle motion-based groove cutting command is specified.
5257	G41/G42 NOT ALLOWED IN MDI MODE	Tool radius/tool nose radius compensation was specified in MDI mode. (Depending on the setting of the bit 4 (MCR) of parameter No. 5008)

Number	Message	Description
5303	TOUCH PANEL ERROR	The touch panel is not connected correctly, or the touch panel cannot be initialized when the power is turned on. Correct the cause then turn on the power again.
5305	ILLEGAL SPINDLE NUMBER	In a spindle select function by address P for a multiple spindle control, 1) Address P is not specified. 2) Parameter No.3781 is not specified to the spindle to be selected. 3) An illegal G code which cannot be commanded with an S_P_; command is specified. 4) A multi spindle cannot be used because the bit 1 (EMS) of parameter No. 3702 is 1. 5) The spindle amplifier number of each spindle is not set in parameter No. 3717. 6) A prohibited command for a spindle was issued (parameter No. 11090). 7) An invalid value is set in parameter No. 11090.
5312	ILLEGAL COMMAND IN G10 L75/76/77	One of formats in G10L75, G10L76, or G10L77 to G11 commands is in error, or the command value is out of data range. Modify the program.
5316	TOOL TYPE NUMBER NOT FOUND	A tool with the specified tool-type number could not be found. Modify the program or register the tool.
5317	ALL TOOL LIFE IS OVER	The lives of all tools with the specified tool-type number have expired. Replace the tool.
5320	DIA./RAD. MODE CAN'T BE SWITCHED .	In any of the following states, diameter/radius specification was switched: 1) When a buffered program is being executed 2) When a movement is being made on the axis
5324	REFERENCE RETURN INCOMPLETE	Manual reference position return cannot be performed during three-dimensional coordinate conversion, execution of the tilted working plane command, or workpiece setting error compensation.
5329	M98 AND NC COMMAND IN SAME BLOCK	A subprogram call which is not a single block was commanded during canned cycle mode.
5339	ILLEGAL FORMAT COMMAND IS EXECUTED IN SYNC/MIX/OVL CONTROL.	1. The value of P, Q, or L specified by G51.4/G50.4/G51.5/G50.5/G51.6/G50.6 is invalid. 2. A duplicate value is specified by parameter No. 12600.

Number	Message	Description
5346	RETURN TO REFERENCE POINT	The coordinate establishment of the Cs contour control axis is not made. Perform a manual reference position return. 1. When Cs coordinate establishment is made for the Cs-axis for which the Cs-axis reference position status signal CSPENx is 0 2. When positional information is not sent from the spindle amplifier 3. When the servo off state is entered during the start of Cs-axis coordinate establishment 4. When the Cs-axis is subjected to synchronous control or superposition control 5. When the emergency stop state is entered during coordinate establishment 6. When an attempt is made to release composite control for the Cs axis being subjected to coordinate establishment 7. When an attempt is made to start synchronous, composite, or superposition control for the Cs axis being subjected to coordinate establishment.
5360	TOOL INTERFERENCE CHECK ERROR	This alarm is issued when interference with another tool is caused by a data modification based on G10 data input or file inputting or when an attempt is made to modify the tool figure data of a tool registered in the cartridge.
5361	ILLEGAL MAGAZINE DATA	Tools stored in the cartridge are interfering with each other. Reregister the tools in the cartridge, or modify the tool management data or tool figure data. If this alarm is issued, no tool interference check is made when tools are registered in the cartridge management table. Moreover, empty pot search operation does not operate normally. If this alarm is issued, the power must be turned off before operation is continued.
5362	CONVERT INCH/MM AT REF-POS	An inch/metric conversion was performed at a position other than the reference position. Perform an inch/metric conversion after returning to the reference position.
5364	ILLEGAL COMMAND IN PROGRAM CHECK	(1) An unspecifiable G code was specified in the high-speed program check mode. (2) The angular axis control option or customer's board option is enabled. (3) One of the following operations was performed. - Chopping in the high-speed program check mode - Starting the high-speed program check mode during chopping - High-speed cycle machining in the high-speed program check mode - Reference position return of an axis for which the reference position is not established, in the high-speed program check mode (4) Switching of PMC axis selection signal EAX*<G0136> was performed. (5) G10 was specified for bit 3 (PGR) of parameter No. 3454 in the high-speed program check mode. (6) G10 was specified for bit 6 (PGS) of parameter No. 3001 in the high-speed program check mode.
5365	NOT CHANGE OF PROGRAM CHECK MODE	Switching of high-speed program check input signal PGCK<Gn290.5> was performed during execution of the program.

Number	Message	Description
5372	IMPROPER MODAL G-CODE (G53.2)	In a block in which G53.2 is specified, a G code in group 01 other than G00 and G01 is specified. Or, G53.2 is specified when the modal G code in group 01 is in a state other than the G00 and G01 states.
5373	ARGUMENT CONVERSION ERROR	For outputting a target MDI program for program restart, a macro call argument cannot be converted to a 9-digit number.
5374	FSC MODE MISMATCH IN RESTART	The current flexible synchronous mode differs from the flexible synchronous mode specified in a programmed command in the program restart block.
5375	FSC MODE CAN NOT CHANGED	The flexible synchronous mode was changed during the execution of program restart.
5376	FSC SLAVE AXIS CAN NOT COMMANDED	In the flexible synchronous mode, a command was specified for the slave axis.
5377	INVALID COMMAND AFTER FSC OFF	After the flexible synchronous mode was canceled, an incremental command was specified before an absolute command for the axis specified as the slave axis.
5378	INVALID RESTART BLOCK	The block specified as the restart block after the flexible synchronous mode was canceled was not a block after an absolute command for the axis specified as the slave axis.
5379	WRITE PROTECTED TO SLAVE AXIS	A parameter cannot be set directly for a slave axis of axis synchronous control.
5381	INVALID COMMAND IN FSC MODE	An attempt was made to issue the following commands: 1 When the reference position for the master axis under flexible synchronization control has not been established, G28 command for the master axis. 2 G27/G28/G29/G30/G30.1/G53 command for a slave axis.
5384	RETRACT FOR RIGID CANNOT BE CMD	In retraction for rigid tapping by the G30 command, different coordinate system modes are used at the stop of rigid tapping and in retraction for rigid tapping.
5391	CAN NOT USE G92	Workpiece coordinate system setting G92 (or G50 for the lathe system G-code system A) cannot be specified. (1) After tool length compensation was changed by tool length compensation shift type, G92 was specified when no absolute command is present. (2) G92 was specified in the block in which G49 is present.
5406	G41.3/G40 FORMAT ERROR	(1) The G41.3 or G40 block contains a move command. (2) The G41.3 block contains a G or M code that suppresses buffering.
5407	ILLEGAL COMMAND IN G41.3	(1) In the G41.3 mode, a G code of group 01 other than G00 and G01 is specified. (2) In the G41.3 mode, an offset command (a G code of group 07) is specified. (3) The block next to G41.3 (startup) specifies no movement.
5408	G41.3 ILLEGAL START_UP	(1) In a mode of group 01 other than G00 and G01, G41.3 (startup) is specified. (2) The included angle between the tool vector and move vector is 0 or 180 degrees at the time of startup.

Number	Message	Description
5420	ILLEGAL PARAMETER IN G43.4/G43.5	A parameter related to tool center point control is illegal. - Acceleration/deceleration before interpolation is disabled. Set parameter No. 1660. - Rapid traverse acceleration/deceleration before interpolation is disabled. Set bit 1 of parameter No. 1401, bit 5 of parameter No. 19501, parameter No. 1671, and parameter No. 1672. - The AI contour control I or AI contour control II option is absent. Set bit 2 (AAI) of parameter No. 11260 to 0.

Number	Message	Description
5421	ILLEGAL COMMAND IN G43.4/G43.5	An illegal command was specified in tool center point control. - A rotation axis command was specified in tool center point control (type 2) mode. - With a table rotary type or mixed-type machine, a I,J,K command was specified in the tool center point control (type 2) command (G43.5) block. - A command that does not move the tool center point (only a rotation axis is moved) was specified for the workpiece in the G02/G03 mode. - When the workpiece coordinate system is set as the programming coordinate system (bit 5 (WKP) of parameter No. 19696 is 1), G02 or G03 was specified while the rotation axis was not perpendicular to the plane. - A G code not specifiable during the tool center point control mode was specified. - The modal code used to specify tool center point control is incorrect. - If, in tool center point control mode, any of the following conditions is met, an axis not related to tool center point control (non 5-axis machining control axis) is specified: (1) The option, the expansion of axis move command in tool center point control, is not provided. (2) The number of non 5-axis machining control axes exceeds the maximum number of axes that can be specified. (3) Nano smoothing or NURBS interpolation is performed. - When bit 0 (RCM) of parameter No. 11200 is set to 0 to disable tool direction compensation, tool center point control is specified during the workpiece setting error compensation/tilted working plane command mode. - When tool posture control is enabled under tool center point control (type 2), a command is specified to set a tool posture near a singular point. (This alarm may be suppressed with bit 3 (NPC) of parameter No. 19696.) Check the machine configuration and specification. - When tool posture control is enabled under tool center point control (type 1), a rotary axis angular displacement that disables tool posture control is specified. Check the machine configuration and specification. - During tool center point control (type 2) or tool posture control, nano smoothing or NURBS interpolation is specified. Check the specification. - For nano smoothing in tool center point control (type 1), only linear axes are specified as axes for nano smoothing. Specify rotation axes. - In a state in which the shift of a mirror image remains, tool center point control, tool posture control, or the cutting point command is specified. - It is intervened with an MDI command during a single block stop in high-speed smooth TCP mode.
5422	EXCESS VELOCITY IN G43.4/G43.5	An attempt was made to make a movement at an axis feedrate exceeding the maximum cutting feedrate by tool center point control.
5424	ILLEGAL TOOL DIRECTION	The rotation axis position for specifying the tool axis direction is not ±90° × n (n = 0, 1, 2,・・・).
5425	ILLEGAL OFFSET VALUE	The offset number is incorrect.

Number	Message	Description
5430	ILLEGAL COMMAND IN 3-D CIR	In a modal state in which 3-dimensional circular interpolation cannot be specified, a 3-dimensional circular interpolation (G02.4/G03.4) is specified. Alternatively, in 3-dimensional circular interpolation mode, a code that cannot be specified is specified.
5432	G02.4/G03.4 FORMAT ERROR	A 3-dimensional circular interpolation command (G02.4/G03.4) is incorrect.
5433	MANUAL INTERVENTION IN G02.4/G03.4 (ABS ON)	In 3-dimensional circular interpolation mode (G02.4/G03.4), manual intervention was made when the manual absolute switch was on.
5435	PARAMETER OUT OF RANGE (TLAC)	Illegal parameter setting. (Set value is out of range.) Check parameters Nos. 19655, 19656, 19657, and 1022.
5436	ILLEGAL PARAMETER SETTING OF ROTARY AXIS(TLAC)	Illegal parameter setting. (axis of rotation setting)
5437	ILLEGAL PARAMETER SETTING OF MASTER ROTARY AXIS(TLAC)	Illegal parameter setting. (master axis of rotation setting)
5445	CAN NOT COMMAND MOTION IN G39	Corner circular interpolation (G39) of tool radius/tool nose radius compensation is not specified alone but is specified with a move command.
5446	NO AVOIDANCE AT G41/G42	Because there is no interference avoidance vector, the interference check avoidance function of tool radius/tool nose radius compensation does not work.
5447	DANGEROUS AVOIDANCE AT G41/G42	The interference check avoidance function of tool radius/tool nose radius compensation operation will lead to danger.
5448	INTERFERENCE TO AVD. AT G41/G42	In the interference check avoidance function of tool radius/tool nose radius compensation, a further interference occurs for an already created interference avoidance vector.
5456	TOO MANY G68.2 NESTING	Tilted working plane command G68.2 was specified more than once. To perform another coordinate conversion, perform cancellation, then specify the coordinate conversion.
5457	G68.2 FORMAT ERROR	A G68.2 format error occurred.
5458	ILLEGAL USE OF G53.1/G53.6	- G53.1/G53.6 was specified preceding G68.2. - G53.1/G53.6 needs to be specified solely. - There is no angle solution for the rotation axis that controls the tool direction in the +Z-axis direction of the feature coordinate system. - In the setting by which compensation in the tool direction is not performed (when bit 0 (RCM) of parameter No. 11200 is 0), G53.1 was specified in the workpiece placement error compensation mode. - No tool is specified with a G53.6 command.

Number	Message	Description
5459	MACHINE PARAMETER INCORRECT	- The parameters Nos.19665 to 19667, 19680 to 19714, and 12321 for configuring the machine are incorrect. - The axis specified with parameter No. 19681 or No. 19686 is not a rotary axis. - In parameter No. 1022, the basic three axes are not set. - A rotary axis end point found by the NC with tool center point control type 2, 3-dimensional cutter compensation type 2, or the tilted working plane command is not within the range set by parameter No. 19741 to No. 19744. - No rotary axis end point was found with tool center point control type 2 or 3-dimensional cutter compensation type 2. Check the machine configuration and specification. - On a machine whose rotary axis is a virtual axis, tool center point control type 2 or 3-dimensional cutter compensation type 2 was specified. - When programming is performed in the workpiece coordinate system, tool center point control type 2 or 3-dimensional cutter compensation type 2 is specified. - If a parameter for setting a 5-axis machining function, the reference rotation axis position, is to be set in the machine coordinate system (bit 7 (SPM) of parameter No. 19754 = 1), an inclined rotary axis is used.

Number	Message	Description
5460	ILLEGAL USE OF 3-DIMENSIONAL CUTTER COMPENSATION	- In the 3-dimensional cutter compensation mode (except the tool side offset function for a tool rotation type machine), a move command other than G00/G01 is specified. - With a table rotation type machine, when bit 1 (PTD) of parameter No. 19746 is set to 1, a plane selection is made with an axis other than the basic three axes at the start of 3-dimensional cutter compensation. - When bit 1 (SPG) of parameter No. 19607 is set to 1, there is a discrepancy between the machine type set in parameter No. 19680 and a G code specifying 3-dimensional cutter compensation (G41.2, G42.2, G41.4, G42.4, G41.5, or G42.5). - With a machine that is not of the tool rotation type, G41.3 is specified. - When bit 5 (WKP) of parameter No. 19696 is set to 0, and bit 4 (TBP) of parameter No. 19746 is set to 0, 3-dimensional cutter compensation and tool center point control are used at the same time. - A rotation axis command is specified in the 3-dimensional cutter compensation mode (type 2). - With a table rotation type or mixed type machine, IJK is specified in a block that specifies 3-dimensional cutter compensation (type 2) (G41.6/G42.6). - An illegal G code is specified in the 3-dimensional cutter compensation mode. - When 3-dimensional cutter compensation is specified, the modal state is illegal. - When the table coordinate system is set as the programming coordinate system, table rotate and 3-dimensional cutter compensation are specified after the start of tool center point control. - There is a difference in type1/type 2 specification between 3-dimensional cutter compensation and tool center point control. - When 3-dimensional cutter compensation and tool center point control are used at the same time, one of these functions that is specified earlier than the other is canceled earlier.
5461	ILLEGAL USE OF G41.2/G42.2/G41.5/G42.5	A move command other than G00 or G01 was performed during 3-dimensional cutter compensation in a mixed-type machine.
5462	ILLEGAL COMMAND (G68.2/G69)	(1) The modal setting used when G68.2 or G69 is specified is incorrect. (2) An unspecifiable G code was specified in the G68.2 mode. (3) The offset vector of tool radius/tool nose radius compensation is not canceled when G68.2 or G69 is specified.
5463	ILLEGAL PARAMETER IN 3-DIMENSIONAL CUTTER COMPENSATION	A parameter related to 3-dimensional cutter compensation is illegal. - Acceleration/deceleration before interpolation is disabled. Set parameter No. 1660. - Rapid traverse acceleration/deceleration before interpolation is disabled. Set bit 1 (LRP) of parameter No. 1401, bit 5 (FRP) of parameter No. 19501, and parameters Nos. No.1671 and 1672.

Number	Message	Description
5464	ILLEGAL COMMAND IN G43.8/G43.9	An illegal value is specified with the cutting point command of tool center point control. - A negative value is set to the parameter (No.11419). - In case of tool compensation memory A and B of machining center system, D code is specified in G43.8/G43.9 block. - In case of tool offset memory C and bit 3 (ON1) of parameter No. 11269 is 1 in machining center system, D code is specified in G43.8/G43.9 block. - All of (I, J, K) following ",L2" are specified as 0. - A value is specified that causes the angle formed by the tool length offset direction and the direction perpendicular to the cutting surface to exceed 90 degrees. - The tool radius offset is less than the corner-R offset. - On a tool rotation type machine, manual intervention is performed on the rotation axis.
5559	ILL. AXIS OPERATION (COORD SYS)	When the parameter MSC(No.11501#2)=1, and the workpiece coordinate system is shifted from the machine coordinate system by the movement command in the machine lock state, the axis movement was commanded. Please operate "reference position return" or "workpiece coordinate system preset" etc.

(4) Parameter writing alarm (SW alarm)

Number	Message	Description
SW0100	PARAMETER ENABLE SWITCH ON	The parameter setting is enabled (bit (PWE) of parameter No. 8000 is set to “1”). To set the parameter, turn this parameter ON. Otherwise, set to OFF.

(5) Servo alarms (SV alarm)

Number	Message	Description
SV0001	SYNC ALIGNMENT ERROR	In feed axis synchronization control, the amount of compensation for synchronization exceeded the parameter No. 8325 setting value. This alarm occurs for a master or slave axis.
SV0002	SYNC EXCESS ERROR ALARM 2	In feed axis synchronization control, the amount of synchronization error exceeded the parameter No. 8332 setting value. When the synchronization is not completed after power-up, the determination is made by the parameter value (No. 8332) multiplied by the parameter No. 8330 multiplier. This alarm occurs only for a slave axis only.
SV0003	SYNCHRONOUS/COMPOSITE/SUPER IMPOSED CONTROL MODE CAN'T BE CONTINUED	Since as axis in synchronization, composition, or superposition mode caused a servo alarm, the mode could not be continued, If one of the axes in a mode causes a servo alarm, all axes relating to the axis enter the servo-off state. This alarm is generated to enable the cause of the servo-off state to be checked.
SV0004	EXCESS ERROR (G31)	The amount of positional deviation during torque limit skip command operation exceeded the limit value of the parameter No.6287.

Number	Message	Description
SV0005	SYNC EXCESS ERROR (MCN)	In feed axis synchronization control, for synchronization, the difference value of the machine coordinate between a master and slave axes exceeded the parameter No. 8314 setting value. This alarm occurs for a master or slave axis.
SV0006	ILLEGAL TANDEM AXIS	For the slave axis under tandem control, absolute position detection is set (parameter bit 5 (APC) of parameter No. 1815 = 1).
SV0007	SV ALM ANOTHER PATH(MULTI AMP.)	When a multi-axis amplifier was used in a multi-path system across paths, a servo alarm occurred on an axis belonging to another path. When a system with two or more paths and multiple servo axes between paths are controlled by a multi-axis amplifier, if a servo alarm occurs on an axis belonging to another path of the same amplifier, the MCC of the amplifier goes down and SV0401 V-READY OFF occurs on an axis belonging to the local path in the same amplifier. Since SV0401 is caused by a servo alarm occurred on an axis in another path, SV0007 is caused together to clearly indicate the fact. The axis belonging to another path in the same amplifier resolves the cause of the servo alarm.
SV0010	SV OVERHEAT	Amplifier internal overheat
SV0011	SV MOTOR OVER CURRENT(SOFT)	The digital servo software detected an abnormal specified value. Possible causes include an unconnected power cable, cable disconnection (open phase), and short-circuit.
SV0012	SV DRIVE OFF CIRCUIT FAILURE	The two drive off inputs are not in the same status or a drive off circuit error occurred.
SV0013	SV CPU BUS FAILURE	An error was found in CPU bus data in the amplifier.
SV0014	SV CPU WATCH DOG	An error occurred in CPU operation in the amplifier.
SV0015	SV LOW VOLT DRIVER	The driver power supply voltage has dropped in the amplifier. Possible causes include improper insertion of the control PC board and amplifier failure.
SV0016	SV CURRENT DETECT ERROR	An error was found in motor current detection data in the amplifier. Possible causes include improper insertion of the control PC board and amplifier failure.
SV0017	SV INTERNAL SERIAL BUS FAILURE	An error occurred in serial bus communication in the amplifier. Possible causes include improper insertion of the control PC board and amplifier failure.
SV0018	SV ROM DATA FAILURE	An error was found in ROM data in the amplifier.
SV0019	PS GROUND FAULT	A ground fault occurred in the motor, power cable, or amplifier.
SV0020	PS GROUND FAULT	A ground fault occurred in the motor, power cable, or amplifier.
SV0021	PS OVERCURRENT 2	Overcurrent flowed in the input circuit.
SV0022	PS OVERCURRENT 3	Overcurrent flowed in the input circuit.
SV0023	PS OVERCURRENT 4	Overcurrent flowed in the input circuit.
SV0024	PS SOFT THERMAL	A load higher than the rating was applied.
SV0025	PS OVER VOLT. DC LINK 2	Overvoltage of the DC link section
SV0026	PS OVER VOLT. DC LINK 3	Overvoltage of the DC link section
SV0027	PS OVER VOLT. DC LINK 4	Overvoltage of the DC link section
SV0028	PS LOW VOLT. DC LINK 2	The DC link voltage has dropped.
SV0029	PS LOW VOLT. DC LINK 3	The DC link voltage has dropped.
SV0030	PS LOW VOLT. DC LINK 4	The DC link voltage has dropped.
SV0031	PS ILLEGAL PARAMETER	An invalid value is set for a PS control parameter.

Number	Message	Description
SV0032	PS CONTROL AXIS ERROR 1	An invalid value is set for parameter No. 2557. Set parameter APS (No.11549#0) to 1 and execute automatic setting.
SV0033	PS CONTROL AXIS ERROR 2	An invalid value is set for parameter No. 2557. Set parameter APS (No.11549#0) to 1 and execute automatic setting.
SV0034	PS HARDWARE ERROR	A PS hardware error was detected.
SV0040	PS SUB MODULE ERROR 1	Failure of PS SUB MODULE, PS or cable. Replace PS SUB MODULE, PS or cable.
SV0041	PS SUB MODULE ERROR 2	Failure of PS SUB MODULE, PS or cable. Replace PS SUB MODULE, PS or cable.
SV0042	PS SUB MODULE ERROR 3	Failure of PS SUB MODULE, PS or cable. Replace PS SUB MODULE, PS or cable.
SV0043	PS SUB MODULE ERROR 4	Failure of PS SUB MODULE, PS or cable. Replace PS SUB MODULE, PS or cable.
SV0044	MISMATCHED FUNCTION CODE	CNC, SV, SP or PS software has been update. Turn the power off, then restart.
SV0301	APC ALARM: COMMUNICATION ERROR	Since the absolute-position detector caused a communication error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0302	APC ALARM: OVER TIME ERROR	Since the absolute-position detector caused an overtime error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0303	APC ALARM: FRAMING ERROR	Since the absolute-position detector caused a framing error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0304	APC ALARM: PARITY ERROR	Since the absolute-position detector caused a parity error, the correct machine position could not be obtained. (data transfer error) The absolute-position detector, cable, or servo interface module is thought to be defective.
SV0305	APC ALARM: PULSE ERROR	Since the absolute-position detector caused a pulse error, the correct machine position could not be obtained. The absolute-position detector, or cable is thought to be defective.
SV0306	APC ALARM: OVER FLOW ERROR	Since the amount of positional deviation overflowed, the correct machine position could not be obtained. Check to see the parameter No. 2084 or No. 2085.
SV0307	APC ALARM: MOVEMENT EXCESS ERROR	Since the machine moved excessively, the correct machine position could not be obtained.
SV0360	ABNORMAL CHECKSUM(INT)	The checksum alarm occurred on the built–in Pulsecoder.
SV0361	ABNORMAL PHASE DATA(INT)	The phase data abnormal alarm occurred on the built–in Pulsecoder.
SV0363	ABNORMAL CLOCK(INT)	The clock alarm occurred on the built–in Pulsecoder.
SV0364	SOFT PHASE ALARM(INT)	A digital servo soft detected an abnormality on the built in Pulsecoder.
SV0365	BROKEN LED(INT)	The digital servo software detected abnormal data on the built–in Pulsecoder.
SV0366	PULSE MISS(INT)	A pulse error occurred on the built–in Pulsecoder.
SV0367	COUNT MISS(INT)	A count error occurred on the built–in Pulsecoder.

Number	Message	Description
SV0368	SERIAL DATA ERROR(INT)	The communications data could not be received from the built–in Pulsecoder.
SV0369	DATA TRANS. ERROR(INT)	A CRC error or stop bit error occurred in the communications data from the built–in Pulsecoder.
SV0380	BROKEN LED(EXT)	Separate detector error
SV0381	ABNORMAL PHASE (EXT)	An abnormal alarm in the position data occurred on the separate detector.
SV0382	COUNT MISS(EXT)	A count error occurred on the separate detector.
SV0383	PULSE MISS(EXT)	A pulse error occurred on the separate detector.
SV0384	SOFT PHASE ALARM(EXT)	The digital servo software detected abnormal data on the separate detector.
SV0385	SERIAL DATA ERROR(EXT)	The communications data could not be received from the separate detector.
SV0386	DATA TRANS. ERROR(EXT)	A CRC error or stop bit error occurred in the communications data from the standalone detector.
SV0387	ABNORMAL ENCODER(EXT)	An abnormality occurred on a separate detector. For more information, contact the scale manufacturer.
SV0401	IMPROPER V_READY OFF	Although the ready signal (PRDY) of the position control was ON, the ready signal (VRDY) of the velocity control was OFF.
SV0403	CARD/SOFT MISMATCH	The combination of the axis control card and the servo software is incorrect. Probable causes are given below. - The correct axis control card is not attached. - The correct servo software is not installed in flash memory.
SV0404	IMPROPER V_READY ON	Although the ready signal (PRDY) of the position control was OFF, the ready signal (VRDY) of the velocity control was ON.
SV0407	EXCESS ERROR	The difference value of the amount of positional deviation for the synchronization axis exceeded the setting value. (during synchronization control only)
SV0409	DETECT ABNORMAL TORQUE	An abnormal load was detected on the servo motor, or during Cs axis or spindle positioning. The alarm can be canceled by RESET.
SV0410	EXCESS ERROR (STOP)	The amount of positional deviation during stopping exceeded the parameter No. 1829 setting value.
SV0411	EXCESS ERROR (MOVING)	The amount of positional deviation during traveling became excessive than the parameter setting value.
SV0413	LSI OVERFLOW	The counter for the amount of positional deviation overflowed
SV0415	MOTION VALUE OVERFLOW	The velocity exceeding the travel velocity limit was commanded.

Number	Message	Description
SV0417	ILL DGTL SERVO PARAMETER	A digital serve parameter setting is incorrect. When bit 4 of diagnosis data No. 203 is 1, an illegal parameter was detected by the servo software. Identify the cause with reference to diagnosis data No. 352. When bit 4 of diagnosis data No. 203 is 0, the CNC software detected an illegal parameter. Probable causes are given below (see diagnosis data No. 280). 1) The value specified in parameter No. 2020 as the motor model falls outside the specified range. 2) The motor rotation direction in parameter No. 2022 is not set to a correct value (111 or -111). 3) The speed feedback pulse count per motor rotation in parameter No. 2023 is set to a negative or other incorrect value. 4) The position feedback pulse count per motor rotation in parameter No. 2024 is set to a negative or other incorrect value.
SV0420	SYNC TORQUE EXCESS	In feed axis synchronization control, for synchronization, the difference value of torque between a master and slave axes exceeded the parameter No. 2031 setting value. This alarm occurs for a master axis.
SV0421	EXCESS ERROR(SEMI-FULL)	The difference between the feedback from the semi and full sides exceeded the setting of parameter No. 2118.
SV0422	EXCESS VELOCITY IN TORQUE	In torque control, the commanded permissible velocity was exceeded.
SV0423	EXCESS ERROR IN TORQUE	In torque control, the total permissible move value specified as a parameter was exceeded.
SV0430	SV MOTOR OVERHEAT	The servo motor has overheated.
SV0431	PS OVERLOAD	Overheat
SV0432	PS LOW VOLT. CONTROL	The control power supply voltage has dropped.
SV0433	PS LOW VOLT. DC LINK	Low DC link voltage
SV0434	SV LOW VOLT CONTROL	Low control power voltage
SV0435	SV LOW VOLT DC LINK	Low DC link voltage
SV0436	SOFTTHERMAL(OVC)	The digital servo software detected a software thermal (OVC).
SV0437	PS OVERCURRENT	Overcurrent on input circuit section.
SV0438	SV ABNORMAL CURRENT	Motor overcurrent
SV0439	PS OVER VOLT. DC LINK	The DC link voltage is too high.
SV0440	PS EXCESS-REGENERATION2	Excessive generative discharge
SV0441	ABNORMAL CURRENT OFFSET	The digital servo software detected an abnormality in the motor current detection circuit.
SV0442	PS PRE-CHARGE FAILURE	The spare charge circuit for the DC link is abnormal.
SV0443	PS INTERNAL FAN FAILURE	Internal cooling fan failure.
SV0444	SV INTERNAL FAN FAILURE	Internal cooling fan failure.
SV0445	SOFT DISCONNECT ALARM	The digital servo software detected a disconnected Pulsecoder.
SV0446	HARD DISCONNECT ALARM	The hardware detected a disconnected built–in Pulsecoder.
SV0447	HARD DISCONNECT(EXT)	The hardware detected a disconnected separate detector.
SV0448	UNMATCHED FEEDBACK ALARM	The sign of the feedback signal from the standalone detector is opposite to that from the feedback signal from the built–on Pulsecoder.
SV0449	SV IPM ALARM	The IPM (Intelligent Power Module) detected an alarm.

Number	Message	Description
SV0453	SPC SOFT DISCONNECT ALARM	Software disconnection alarm of the built-in Pulsecoder. Turn off the power to the CNC, then remove and insert the Pulsecoder cable. If this alarm is issued again, replace the Pulsecoder.
SV0454	ILLEGAL ROTOR POS DETECT	The magnetic pole detection function terminated abnormally. The magnetic pole could not be detected because the motor did not run.
SV0456	ILLEGAL CURRENT LOOP	An attempt was made to set the current loop that could not be set. The amplifier pulse module in use does not comply with HIGH SPEED HRV. Or, requirements to control are not satisfied in the system.
SV0458	CURRENT LOOP ERROR	The specified current loop differs from the actual current loop.
SV0459	HI HRV SETTING ERROR	For two axes whose servo axis numbers (parameter No. 1023) are consecutively even and odd numbers, HIGH SPEED HRV control is possible for one axis and impossible for the other.
SV0460	FSSB DISCONNECT	The FSSB connection was discontinued. Probable causes are: 1. The FSSB connection cable was disconnected or broken. 2. The amplifier was turned off . 3. In the amplifier, the low-voltage alarm occurred.
SV0462	SEND CNC DATA FAILED	The correct data could not be received on a slave side because of the FSSB communication error.
SV0463	SEND SLAVE DATA FAILED	The correct data could not be received in the servo software because of the FSSB communication error.
SV0465	READ ID DATA FAILED	A read of the ID information for the amplifier has failed at power-on.
SV0466	MOTOR/AMP. COMBINATION	The maximum current of an amplifier is different to that of a motor. Probable causes are: 1. The connection command for an amplifier is incorrect. 2. The parameter No.2165 setting is incorrect
SV0468	HI HRV SETTING ERROR(AMP)	An attempt was made to set up HIGH SPEED HRV control for use when the controlled axis of an amplifier for which HIGH SPEED HRV control could not be used.
SV0474	EXCESS ERROR(STOP:SV)	The servo detected that the positional deviation during a stop exceeded the setting (parameters Nos. 1839 and 1842) in the n-axis.
SV0475	EXCESS ERROR(MOVE:SV)	The servo detected that the positional deviation during a travel exceeded the setting (parameters Nos. 1838 and 1841) in the n-axis.
SV0476	ILLEGAL SPEED CMD.(SV)	The servo detected that the speed command exceeded the safety speed (parameters Nos. 13821 to 13824 (during position control) or parameters Nos. 13826 to 13829 (during speed control)) during safety monitoring (when safety monitoring request signal *VLDVx is 0) in the n-axis. Keep the safety speed.

Number	Message	Description
SV0477	ILLEGAL MACHINE POS.(SV)	The servo detected that the machine position fell outside the safety area (setting by parameters Nos. 13831 to 13838) during safety monitoring (when safety monitoring request signal *VLDVx is 0) in the n-axis. Keep the safety area. A machine position check is performed only on the axis for which the reference position has been established. The axis for which the reference position is not established is not subjected to a machine position check.
SV0478	ILLEGAL AXIS DATA(SV)	The servo detected that an error occurred during transfer of axis data in the n-axis. When an alarm occurred because the configuration of the servo amplifier was changed, set the axis number for the servo amplifier (set bit 4 of parameter No. 2212 of the corresponding axis to 1 and then 0 and turn off the power of the entire system). When using a multiaxis amplifier, this operation may not clear the alarm. In this case, repeat this operation for the axes on which the alarm persists. If an alarm occurs due to a cause other than the above, replace the servo amplifier.
SV0481	SAFETY PARAM ERROR(SV)	The servo detected that a safety parameter error occurred in the n-axis.
SV0484	SAFETY FUNCTION ERROR(SV)	A safety function error related to the servo was detected in the n-axis. - The servo or CNC detected that the safety function was not executed in the servo. - The result of a servo safety function check did not match the result of a CNC safety function check. - An error occurred during a test of the CPU of the servo. - An error occurred during a check of RAM of the servo.
SV0488	SELF TEST OVER TIME	An MCC interruption test was not complete within the set period of time (parameter No. 1946). Check the MCC contact.
SV0489	SAFETY PARAM ERROR(CNC)	The CNC detected that a safety parameter error occurred in the n-axis.
SV0490	SAFETY FUNCTION ERROR(CNC)	A CNC safety function error occurred in the n-axis. - The servo detected that the safety function was not executed in the CNC. - The result of a servo safety function check did not match the result of a CNC safety function check.
SV0494	ILLEGAL SPEED CMD.(CNC)	The CNC detected that the speed command exceeded the safety speed (parameters Nos. 13821 to 13824 (during position control) or parameters Nos. 13826 to 13829 (during speed control)) during safety monitoring (when safety monitoring request signal *VLDVx is 0) in the n-axis. Keep the safety speed.
SV0495	ILLEGAL MACHINE POS.(CNC)	The CNC detected that the machine position fell outside the safety area (set range by parameters Nos. 13831 to 13838) during safety monitoring (when safety monitoring request signal *VLDVx is 0) in the n-axis. Keep the safety area. A machine position check is performed only on the axis for which the reference position has been established.

Number	Message	Description
SV0496	ILLEGAL AXIS DATA(CNC)	The CNC detected that an error occurred during transfer to axis data. When an alarm occurred because the configuration of the servo amplifier was changed, set the axis number for the servo amplifier (set bit 4 of parameter No. 2212 of the corresponding axis to 1 and 0 again and turn off the power of the entire system). When using a multiaxis amplifier, this operation may not clear the alarm. In this case, repeat this operation for the axes on which the alarm persists. If an alarm occurs due to a cause other than the above, replace the servo amplifier.
SV0498	AXIS NUMBER NOT SET(CNC)	The CNC detected that the axis number of the n-axis was not set for the servo amplifier. The axis number is set automatically, so turn off the power of the entire system.
SV0600	SV DC LINK OVER CURRENT	DC link overcurrent.
SV0601	SV EXTERNAL FAN FAILURE	Radiator cooling fan failure.
SV0602	SV RADIATOR OVERHEAT	The servo amplifier radiator has overheated.
SV0603	INV. IPM ALARM(OH)	The IPM (Intelligent Power Module) detected an overheat alarm.
SV0604	AMP COMMUNICATION ERROR	The communication between Servo Amplifier (SV) and Common Power Supply (PS) is in error.
SV0605	PS EXCESS-REGENERATION1	The motor regenerative power is too much.
SV0606	PS EXTERNAL FAN FAILURE	External radiator cooling fan failure.
SV0607	PS IMPROPER INPUT POWER	An abnormality was found with the input power supply.
SV0646	ABNORMAL ANALOG SIGNAL(EXT)	An error occurred in the analog 1Vp-p output of the separate detector. The separate detector, cable, or separate detector interface unit may be failed.
SV0652	TEMP.ERROR	Communication between the separate detector interface unit and temperature sensor was disconnected.
SV0654	DB RELAY FAILURE	A failure occurs in the dynamic brake relay of the servo amplifier. Replace the amplifier.
SV1025	V_READY ON (INITIALIZING)	The ready signal (VRDY) of the velocity control which should be OFF is ON while the servo control is ON.
SV1026	ILLEGAL AXIS ARRANGE	The parameter for servo axis arrange is not set correctly. - Parameter No. 1023 (servo axis number of each axis) is set to a negative value or a duplicate value. - The settings for parameter No. 1023 (servo axis number of each axis) were made with a certain setting skipped among 1 to 6, 9 to 14, or 17 to 22. - A setting of a multiple of 8 or a multiple of 8 minus 1 was made.
SV1055	ILLEGAL TANDEM AXIS	In tandem control, the setting of the parameter No. 1023 is incorrect.
SV1067	FSSB:CONFIGURATION ERROR(SOFT)	An FSSB configuration error occurred (detected by software). The connected amplifier type is incompatible with the FSSB setting value.
SV1068	DUAL CHECK SAFETY ALARM	An alarm that turns off the MCC of the entire system occurred in the Dual Check Safety function.
SV1069	EXCESS ERROR(SERVO OFF:CNC)	The CNC detected that the positional deviation during servo-off exceeded the set value (parameter No. 1840) in the n-axis.
SV1070	EXCESS ERROR(SERVO OFF:SV)	The servo detected that the positional deviation during servo-off exceeded the set value (parameter No. 1840) in the n-axis.

Number	Message	Description
SV1071	EXCESS ERROR(MOVE:CNC)	The CNC detected that the positional deviation during a travel exceeded the set value (parameters Nos. 1838 and 1841) in the n-axis.
SV1072	EXCESS ERROR(STOP:CNC)	The CNC detected that the positional deviation during a stop exceeded the set value (parameters Nos. 1839 and 1842) in the n-axis.
SV1100	S-COMP. VALUE OVERFLOW	The amount of compensation for the straightness exceeded a maximum value of 32767.
SV5134	FSSB:OPEN READY TIME OUT	In the initialization, the FSSB could not be in an open ready sate. The axis card is thought to be defective.
SV5136	FSSB:NUMBER OF AMP. IS INSUFFICIENT	The number of amplifier identified by the FSSB is insufficient than the number of control axes. Or, the setting of the number of axes or the amplifier connection is in error.
SV5137	FSSB:CONFIGURATION ERROR	An FSSB configuration error occurred. The connecting amplifier type is incompatible with the FSSB setting value.
SV5139	FSSB:ERROR	Servo initialization has not completed successfully. It is probable that an optical cable failed or a connection between the amplifier and another module failed.
SV5197	FSSB:OPEN TIME OUT	The FSSB could not be opened although the CNC permitted the opening of the FSSB. Check the connection between the CNC and the amplifier.
SV5311	FSSB:ILLEGAL CONNECTION	Different current loops (HRV) are set for FSSB lines. Specify the same current loop for the FSSB lines.

(6) Overtravel alarms (OT alarm)

Number	Message	Description
OT0500	+ OVERTRAVEL (SOFT 1)	Exceeded the positive side stored stroke check 1.
OT0501	- OVERTRAVEL (SOFT 1)	Exceeded the negative side stored stroke check 1.
OT0502	+ OVERTRAVEL (SOFT 2)	Exceeded the positive side stored stroke check 2. Or, in the chuck tail stock barrier, an entry to the inhibited area was made during movement in the positive direction.
OT0503	- OVERTRAVEL (SOFT 2)	Exceeded the negative side stored stroke check 2. Or, in the chuck tail stock barrier, an entry to the inhibited area was made during movement in the negative direction.
OT0504	+ OVERTRAVEL (SOFT 3)	Exceeded the positive side stored stroke check 3.
OT0505	- OVERTRAVEL (SOFT 3)	Exceeded the - side stored stroke check 3.
OT0506	+ OVERTRAVEL (HARD)	The stroke limit switch in the positive direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is not generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.
OT0507	- OVERTRAVEL (HARD)	The stroke limit switch in the negative direction was triggered. This alarm is generated when the machine reaches the stroke end. When this alarm is not generated, feed of all axes is stopped during automatic operation. During manual operation, only the feed of the axis on which the alarm occurred is stopped.

Number	Message	Description
OT0508	INTERFERENCE:+	A tool moving in the positive direction along the n axis has fouled another tool post.
OT0509	INTERFERENCE:-	A tool moving in the negative direction along the n axis has fouled another tool post.
OT0510	+ OVERTRAVEL (PRE-CHECK)	The tool exceeded the limit in the negative direction during the stroke check before movement.
OT0511	- OVERTRAVEL (PRE-CHECK)	The tool exceeded the limit in the positive direction during the stroke check before movement.
OT0514	(n) INTERFERENCE:+	An interference occurred on the plus side of the n-axis in a rotation area interference check.
OT0515	(n) INTERFERENCE:-	An interference occurred on the minus side of the n-axis in a rotation area interference check.
OT0518	(n)+: INTERFERE ([Target name 1],[Target name 2])	A check made for interference between [Target name1] and [Target name2] detected interference on the + side of axis n. n represents the name of the axis with which an alarm is issued.
OT0519	(n)-: INTERFERE ([Target name 1],[Target name 2])	A check made for interference between [Target name1] and [Target name2] detected interference on the - side of axis n. n represents the name of the axis with which an alarm is issued.
OT1710	ILLEGAL ACC. PARAMETER(OPTIMUM TORQUE ACC/DEC)	The permissible acceleration parameter for the optimum torque acceleration/deceleration is in error. A possible cause is either of the following: (1) The ratio of a negative acceleration to a positive acceleration is not more than the limit value. (2) The time to reduce to a velocity of 0 exceeded the maximum time.

(7) Memory file alarms (IO alarm)

Number	Message	Description
IO1001	FILE ACCESS ERROR	The resident–type file system could not be accessed as an error occurred in the resident–type file system.
IO1002	FILE SYSTEM ERROR	The file could not be accessed as an error occurred in the CNC file system.
IO1030	CHECK SUM ERROR	The checksum of the CNC part program storage memory is incorrect.
IO1032	MEMORY ACCESS OVER RANGE	Accessing of data occurred outside the CNC part program storage memory range.
IO1034	PROGRAM FOLDER FILE IS BROKEN	Abnormality of data was detected in the program folder file. It is necessary to initialize the program file to recover. Please refer to the paragraph of the maintenance manual "IPL monitor" for the initialization operation of the program file.
IO1035	PROGRAM MANAGEMENT FILE IS BROKEN	Abnormality of data was detected in the program management file. It is necessary to initialize the program file to recover. Please refer to the paragraph of the maintenance manual "IPL monitor" for the initialization operation of the program file.
IO1104	OVER MAXIMUM TOOL LIFE PAIRS	The maximum number of tool life management pairs is exceeded. Modify the setting of the maximum number of tool life management pairs in parameter No. 6813.

(8) Alarms requiring power to be turned off (PW alarm)

Number	Message	Description
PW0000	POWER MUST BE OFF	A parameter was set for which the power must be turned OFF then ON again.

Number	Message	Description
PW0001	X-ADDRESS(*DEC) IS NOT ASSIGNED.	The X address of the PMC could not be assigned correctly. This alarm may occur in the following case: - During the setting of parameter No. 3013, the X address could not be assigned correctly for the deceleration dog (*DEC) for a return to the reference position.
PW0002	PMC address is not correct(AXIS).	The address to assign the axis signal is incorrect. This alarm may occur in the following case: - The parameter No.3021 setting is incorrect.
PW0003	PMC address is not correct(SPINDLE).	The address to assign the spindle signal is incorrect. This alarm may occur in the following case: - The parameter No.3022 setting is incorrect.
PW0004	SETTING THE LOADER SYSTEM PATH IS NOT CORRECT.	The loader system could not be assigned correctly. The parameter No. 984 setting is incorrect. - The number of loader systems and the number of systems specified to the loader system in the bit 0 (LCP) of parameter No. 0984 does not match. - The bit 0 (LCP) of parameter No. 0984 of the system 1 is set to 1.
PW0006	POWER MUST BE OFF (ILL-EXEC-CHK)	The malfunction prevention function detected an alarm to require the power off.
PW0007	X-ADDRESS(SKIP) IS NOT ASSIGNED	The X address of PMC could not be assigned correctly. Possible causes are: - During the set of parameter No. 3012, the skip signal of the X address was not assigned correctly. - During the set of parameter No. 3019, the address other than the skip signal of the X address was not assigned correctly.
PW0008	CPU SELF TEST ERROR(DCS PMC)	On the DCS PMC side: - An error was detected by the CPU self diagnosis function. - An error was detected by the RAM check function.
PW0009	CPU SELF TEST ERROR(PMC)	On the PMC side: - An error was detected by the CPU self diagnosis function. - An error was detected by the RAM check function.
PW0010	SAFE I/O CROSS CHECK ERROR(DCS PMC)	On the DCS PMC side, an error was detected by the I/O cross check function in system-defined safety-related DI/DO.
PW0011	SAFE I/O CROSS CHECK ERROR(PMC)	On the PMC side, an error was detected by the I/O cross check function in system-defined safety-related DI/DO.
PW0012	USER I/O CROSS CHECK ERROR(DCS PMC)	On the DCS PMC side, an error was detected by the I/O cross check function in user-defined safety-related DI/DO.
PW0013	USER I/O CROSS CHECK ERROR(PMC)	On the PMC side, an error was detected by the I/O cross check function in user-defined safety-related DI/DO.
PW0014	CPU TEST ALARM (CNC)	An error occurred in a test of the CPU of the CNC.
PW0015	SAFETY PARAM ERROR	The CNC detected that an error occurred in a safety parameter for other than servo axes or spindle axes.
PW0016	RAM CHECK ERROR	An error was detected in a RAM check of the CNC.
PW0017	INEXECUTION OF SAFETY FUNCTIONS	The safety function was not executed normally in the CNC.
PW0018	CRC CHECK ERROR	An error was detected in a CRC check of the CNC.

Number	Message	Description
PW0036	ILLEGAL SETTING FOR SERVO MOTOR SPINDLE	The parameter setting for the servo axes for spindle use of the spindle control with servo motor is illegal. - The servo axes number for spindle use has exceeded 4 axes. - Servo axes number for spindle use are fewer than the axes number of designation of servo axes for spindle use. - The servo axes for spindle use is not set as a spindle control with servo motor axis. - The servo axes for spindle use is not set as a rotation axis.. - The servo axis number (parameter No.1023) is a negative value.
PW0050	POWER MUST BE OFF (INITIALIZED COMMUNICATION)	When the power is turned on, the hardware of the communication function was initialized. The power must be turned off to be effective. This alarm may occur in the following cases: - When the option board is added or replaced - When the software option is changed
PW0060	PS CONTROL AXIS ERROR 3	Amplifier group number duplication. Set parameter APS (No.11549#0) to 1 and execute automatic setting.
PW1102	ILLEGAL PARAMETER (I-COMP.)	The parameter for setting slope compensation is incorrect. This alarm occurs in the following cases: - When the number of pitch error compensation points on the axis on which slope compensation is executed exceeds 128 between the most negative side and most positive side - When the size relationship between the slope compensation point Nos. is incorrect - When the slope compensation point is not located between the most negative side and most positive side of pitch error compensation - When the compensation per compensation point is too small or too great.
PW1103	ILLEGAL PARAMETER (S-COMP.128)	The parameter for setting 128 straightness compensation points or the parameter compensation data is incorrect,
PW1104	ILLEGAL PARAMETER (3-D MACHINE POSITION COMPENSATION.)	A parameter for setting 3-dimensional machine position compensation is incorrect.
PW1105	ILLEGAL PARAMETER (3DR-COMP)	Parameter setting for three-dimensional rotary error compensation is wrong. One of the following may be the cause. - The designation for rotary axes is wrong. - The number of the total compensation points is larger than 7812. - The designation for machine type is wrong. - The designation for compensation axis is wrong. - The setting for compensation number of reference point is wrong. - The setting for compensation interval is wrong.

Number	Message	Description
PW5046	ILLEGAL PARAMETER (S-COMP.)	The setting of a parameter related to straightness compensation is invalid. Possible causes are: - A nonexistent axis number is set in a moving or compensation axis parameter. - More than 128 pitch error compensation points are set between the furthest points in the negative and positive regions. - The straightness compensation point numbers do not have correct magnitude relationships. - No straightness compensation point is found between the furthest pitch error compensation point in the negative region and that in the positive region. - The compensation per compensation point is either too large or too small.
PW5390	R-ADDRESS SETTING IS ILLEGAL	The start address or the set range of the PMC R addresses set by parameters Nos.13541 and 13542 is invalid.

(9) Spindle alarms (SP alarm)

Number	Message	Description
SP0740	RIGID TAP ALARM : EXCESS ERROR	The positional deviation of the stopped spindle has exceeded the set value during rigid tapping.
SP0741	RIGID TAP ALARM : EXCESS ERROR	The positional deviation of the moving spindle has exceeded the set value during rigid tapping. The synchronous error range has exceeded the set value.
SP0742	RIGID TAP ALARM : LSI OVERFLOW	An LSI overflow has occurred for the spindle during rigid tapping.
SP0752	SPINDLE MODE CHANGE ERROR	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contour control, spindle positioning, rigid tapping, and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the NC.
SP0754	ABNORMAL TORQUE	An abnormal load was detected in a spindle motor. The alarm can be canceled by RESET.
SP0755	SAFETY FUNCTION ERROR	The CNC CPU detected that the safely function of the n-th spindle was not executed. Alternatively, the result of a CNC safety function check did not match the result of a spindle safety function check.
SP0756	ILLEGAL AXIS DATA	The CNC CPU detected an error during transfer of spindle data in the nth spindle. When this alarm is issued because the configuration of the spindle amplifiers was changed, set the spindle number for the spindle amplifier (set bit 7 of parameter No. 4541 to 1 and then 0, and turn the power to the entire system off).
SP0757	SAFETY SPEED OVER	The CNC CPU detected that during safety monitoring (when safety monitoring request signal *VLDPs is 0), the spindle motor speed was greater than the safety speed (parameter No. 4372, 4438, 4440, or 4442) on the n-th spindle. Operate within the safety speed.
SP1202	SPINDLE SELECT ERROR	In a multi spindle control, the spindle number other than the valid spindle number was selected by a position coder select signal. An attempt was made to select the spindle number of the system having no valid spindle.
SP1210	TOOL CHANGE SP MOTION OVERFLOW	The amount of distribution to a spindle is too much. (specific to the FANUC ROBODRILL)

Number	Message	Description
SP1211	TOOL CHANGE SP ORTN EXCESS ERROR	During a tool change, a too much orientation error was detected for the spindle. (specific to the FANUC ROBODRILL)
SP1212	TOOL CHANGE SP MOVE EXCESS ERROR	During a tool change, a too much moving error was detected for the spindle. (specific to the FANUC ROBODRILL)
SP1213	TOOL CHANGE SP STOP EXCESS ERROR	During a tool change, a too much stop error was detected for the spindle. (specific to the FANUC ROBODRILL)
SP1214	TOOL CHANGE SP ILLEGAL SEQUENCE	During changing tools, an abnormal spindle sequence was detected. (specific to the FANUC ROBODRILL)
SP1220	NO SPINDLE AMP.	Either the cable connected to a serial spindle amplifier is broken, or the serial spindle amplifier is not connected.
SP1221	ILLEGAL MOTOR NUMBER	The spindle No. and the motor No. are incorrectly matched.
SP1224	ILLEGAL SPINDLE-POSITION CODER GEAR RATIO	The spindle–position coder gear ratio was incorrect.
SP1225	CRC ERROR (SERIAL SPINDLE)	A CRC error (communications error) occurred in communications between the CNC and the serial spindle amplifier.
SP1226	FRAMING ERROR (SERIAL SPINDLE)	A framing error occurred in communications between the CNC and the serial spindle amplifier.
SP1227	RECEIVING ERROR (SERIAL SPINDLE)	A receive error occurred in communications between the CNC and the serial spindle amplifier.
SP1228	COMMUNICATION ERROR (SERIAL SPINDLE)	A communications error occurred between the CNC and the serial spindle amplifier.
SP1229	COMMUNICATION ERROR SERIAL SPINDLE AMP.	A communications error occurred between serial spindle amplifiers (motor Nos. 1 and 2, or motor Nos. 3–4).
SP1231	SPINDLE EXCESS ERROR (MOVING)	The position deviation during spindle rotation was greater than the value set in parameters.
SP1232	SPINDLE EXCESS ERROR (STOP)	The position deviation during spindle stop was greater than the value set in parameters.
SP1233	POSITION CODER OVERFLOW	The error counter/speed instruction value of the position coder overflowed.
SP1234	GRID SHIFT OVERFLOW	Grid shift overflowed.
SP1240	DISCONNECT POSITION CODER	The analog spindle position coder is broken.
SP1241	D/A CONVERTER ERROR	The D/A converter for controlling analog spindles is erroneous.
SP1243	ILLEGAL SPINDLE PARAMETER SETTING(GAIN)	The setting for the spindle position gain is incorrect.
SP1244	MOTION VALUE OVERFLOW	The amount of distribution to a spindle is too much
SP1245	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC.
SP1246	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC.
SP1247	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC.
SP1252	ILLEGAL SPINDLE PARAMETER SETTING (TANDEM)	Setting of parameter No.4597 is invalid. Possible causes are : - A value that is larger than the maximum number of controlled spindle axes is set in parameter No.4597. - A negative value except for –1 is set in parameter No.4597. - Relationship of master axis and slave axis that is set in parameter No.4597 is illegal. - When the spindle amplifier that is not applicable the function is used, a value except for 0 is set in parameter No.4597.

Number	Message	Description
SP1700	SAFETY PARAM ERROR	The CNC detected that a safety parameter error occurred in the n-th spindle.
SP1969	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1970	SPINDLE CONTROL ERROR	Initialization of spindle control ended in error.
SP1971	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1972	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1974	ANALOG SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1975	ANALOG SPINDLE CONTROL ERROR	An position coder error was detected on the analog spindle.
SP1976	SERIAL SPINDLE COMMUNICATION ERROR	The amplifier No. could not be set to the serial spindle amplifier.
SP1977	SERIAL SPINDLE COMMUNICATION ERROR	An error occurred in the spindle control software.
SP1978	SERIAL SPINDLE COMMUNICATION ERROR	A time–out was detected during communications with the serial spindle amplifier.
SP1979	SERIAL SPINDLE COMMUNICATION ERROR	The communications sequence was no longer correct during communications with the serial spindle amplifier.
SP1980	SERIAL SPINDLE AMP. ERROR	Defective SIC–LSI on serial spindle amplifier
SP1981	SERIAL SPINDLE AMP. ERROR	An error occurred during reading of the data from SIC–LSI on the analog spindle amplifier side.
SP1982	SERIAL SPINDLE AMP. ERROR	An error occurred during reading of the data from SIC–LSI on the serial spindle amplifier side.
SP1983	SERIAL SPINDLE AMP. ERROR	Could not clear on the spindle amplifier side.
SP1984	SERIAL SPINDLE AMP. ERROR	An error occurred during re–initialization of the spindle amplifier.
SP1985	SERIAL SPINDLE CONTROL ERROR	Failed to automatically set parameters
SP1986	SERIAL SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1987	SERIAL SPINDLE CONTROL ERROR	Defective SIC–LSI on the CNC
SP1988	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1989	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1996	ILLEGAL SPINDLE PARAMETER SETTING	The spindle was assigned incorrectly. Check to see the following parameter. (No.3716 or 3717)
SP1998	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.
SP1999	SPINDLE CONTROL ERROR	An error occurred in the spindle control software.

(10) Alarm list (serial spindle)

When a serial spindle alarm occurs, the following number is displayed on the CNC.

NOTE

*1 Note that the meanings of the Spindle Amplifier (SP) indications differ depending on which LED, the red or yellow LED, is on. When the red LED is on, the Spindle Amplifier (SP) indicates a 2-digit alarm number. When the yellow LED is on, the Spindle Amplifier (SP) indicates an error number that designates a sequence problem (for example, when a rotation command is entered with the emergency stop state not released).
See "Error Codes (Serial Spindle)."

*2 For information about serial spindle alarms for numbers that are not listed below, refer to the following documents according to the actual spindle motor to be connected.
- FANUC AC SPINDLE MOTOR αi series MAINTENANCE MANUAL (B-65285EN)
- Technical reports etc.

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9001	MOTOR OVERHEAT	01	1 Check and correct the peripheral temperature and load status. 2 If the cooling fan stops, replace it.	The internal temperature of the motor exceeds the specified level. The motor is used in excess of the continuous rating, or the cooling component is abnormal.
SP9002	EX DEVIATION SPEED	02	1 Check and correct the cutting conditions to decrease the load. 2 Correct parameter No. 4082.	The motor speed cannot follow a specified speed. An excessive motor load torque is detected. The acceleration/deceleration time in parameter No. 4082 is insufficient.
SP9003	DC-LINK FUSE IS BROKEN	03	1 Replace the Spindle Amplifier (SP). 2 Check the motor insulation status.	The Power Supply becomes ready (00 is indicated), but the DC link voltage is too low in the Spindle Amplifier (SP). The fuse in the DC link section in SP is blown. (The power device is damaged or the motor is ground-fault.)
SP9004	PS IMPROPER INPUT POWER		Check the state of the input power supply to the Power Supply (PS).	The power supply (PS) detected a power supply failure. (Power Supply alarm 14)
SP9006	THERMAL SENSOR DISCONNECT	06	1 Check and correct the parameter. 2 Replace the feedback cable.	The temperature sensor of the motor is disconnected.
SP9007	OVER SPEED	07	Check for a sequence error. (For example, check whether spindle synchronization was specified when the spindle could not be turned.)	The motor speed has exceeded 115% of its rated speed. When the spindle axis was in position control mode, position error were accumulated excessively (SFR and SRV were turned off during spindle synchronization.)
SP9009	OVERHEAT MAIN CIRCUIT	09	1 Improve the heat sink cooling status. 2 If the heat sink cooling fan stops, replace the Spindle Amplifier (SP).	The temperature in the power semiconductor cooling radiator is abnormally high.
SP9010	LOW VOLT INPUT POWER	10	1 Replace the cables. 2 Replace the SP control printed circuit board.	A drop in the input power voltage in the Spindle Amplifier (SP) is detected.
SP9011	PS OVER VOLT. DC LINK		1 Check the selected Power Supply (PS). 2 Check the input power voltage and change in power during motor deceleration. If the voltage exceeds 253 VAC (for the 200-V system) or 530 VAC (for the 400-V system), improve the power supply impedance.	Overvoltage of the DC link section of the Power Supply (PS) was detected. (Power Supply alarm indication: 07) PS selection error. (The maximum output specification of the PS is exceeded.)

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9012	OVERCURRENT POWER CIRCUIT	12	1 Check the motor insulation status. 2 Check the spindle parameters. 3 Replace the Spindle Amplifier (SP).	The motor current is abnormally high. A motor-specific parameter does not match the motor model. Poor motor insulation
SP9013	CPU DATA MEMORY FAULT	13	Replace the Spindle Amplifier (SP) control printed circuit board.	Abnormality in a Spindle Amplifier (SP) control circuit component is detected. (RAM within the Spindle Amplifier (SP) is abnormal.)
SP9014	SOFTWARE SERIES MISMATCH	14	Replace the spindle software.	A Spindle Amplifier (SP) that is not registered in the spindle software is used.
SP9015	SPINDLE SWITCHING FAULT	15	1 Check and correct the ladder sequence. 2 Replace the switching MCC.	The switch sequence in spindle switching/speed range switching operation is abnormal. The switching MCC contact status check signal and command do not match.
SP9016	RAM ERROR	16	Replace the Spindle Amplifier (SP) control printed circuit board.	Abnormality in a Spindle Amplifier (SP) control circuit component is detected. (RAM for external data is abnormal.)
SP9017	ID NUMBER PARITY ERROR	17	Replace the Spindle Amplifier (SP)	Abnormality in Spindle Amplifier (SP) ID data is detected.
SP9018	SUMCHECK ERROR PROGRAM ROM	18	Replace the Spindle Amplifier (SP) control printed circuit board.	Abnormality in a Spindle Amplifier (SP) control circuit component is detected. (Program ROM data is abnormal.)
SP9019	EXCESS OFFSET CURRENT U	19	Replace the Spindle Amplifier (SP).	Abnormality in a Spindle Amplifier (SP) component is detected. (The initial value for the U phase current detection circuit is abnormal.)
SP9020	EXCESS OFFSET CURRENT V	20	Replace the Spindle Amplifier (SP).	Abnormality in a Spindle Amplifier (SP) component is detected. (The initial value of the V phase current detection circuit is abnormal.)
SP9021	POS SENSOR POLARITY ERROR	21	Check and correct the parameters. (parameters Nos. 4000#0, 4001#4)	The polarity parameter setting of the position sensor is wrong.
SP9022	SP AMP OVER CURRENT	22	1 Review operation conditions (acceleration/ deceleration and cutting) to reduce the load. 2 Check and correct the parameters.	A Spindle Amplifier (SP) overload current was detected.
SP9024	SERIAL TRANSFER ERROR	24	1 Place the CNC-to-spindle cable away from the power cable. 2 Replace the cable.	The CNC power is turned off (normal power-off or broken cable). An error is detected in communication data transferred to the CNC.

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9027	DISCONNECT POSITION CODER	27	Replace the cable.	The spindle position coder (connector JYA3) signal is abnormal.
SP9029	OVERLOAD	29	Check and correct the load status.	Excessive load has been applied continuously for a certain period of time. (This alarm is issued also when the motor shaft has been locked in the excitation state.)
SP9030	OVERCURRENT		Check and correct the power supply voltage.	Overcurrent is detected in Power Supply (PS) main circuit input. (Power Supply alarm indication: 01) Unbalanced power supply. PS selection error (The maximum PS output specification is exceeded.)
SP9031	MOTOR LOCK	31	1 Check and correct the load status. 2 Replace the motor sensor cable (connector JYA2).	The motor cannot rotate at a specified speed. (A level not exceeding the SST level for the rotation command has existed continuously.)
SP9032	SIC-LSI RAM FAULT	32	Replace the Spindle Amplifier (SP) control printed circuit board.	Abnormality in a Spindle Amplifier (SP) control circuit component is detected. (The LSI device for serial transfer is abnormal.)
SP9033	PS PRE-CHARGE FAILURE		1 Check and correct the power supply voltage. 2 Replace the Power Supply (PS).	Charging of direct current power supply voltage in the power circuit section is insufficient when the magnetic contractor is turned on (such as open phase and defective charging resistor). (Power Supply alarm indication: 05)
SP9034	ILLEGAL PARAMETER	34	Correct a parameter value according to FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN). If the parameter number is unknown, connect the spindle check board, and check the indicated parameter.	Parameter data exceeding the allowable limit is set.
SP9036	OVERFLOW ERROR COUNTER	36	Check whether the position gain value is too large, and correct the value.	An error counter overflow occurred.
SP9037	ILLEGAL SETTING VELOCITY DETECTOR	37	Correct the parameter value according to FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN).	The setting of the parameter for the number of pulses in the speed detector is incorrect.
SP9041	ILLEGAL 1REV SIGN OF POSITION CODER	41	1 Check and correct the parameter. 2 Replace the cable.	1 The 1-rotation signal of the spindle position coder (connector JYA3) is abnormal. 2 Parameter setting error

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9042	NO 1REV SIGN OF POSITION CODER	42	Replace the cable.	The 1-rotation signal of the spindle position coder (connector JYA3) is disconnected.
SP9043	DISCONNECT POSITION CODER DEF. SPEED	43	Replace the cable.	The position coder signal (connector JYA3S) for differential speed in the submodule SW is abnormal.
SP9046	ILLEGAL 1REV SIGN OF SCREW CUT	46	1 Check and correct the parameter. 2 Replace the cable. 3 Re-adjust the BZ sensor signal.	The 1-rotation signal in threading is abnormal.
SP9047	ILLEGAL SIGNAL OF POSITION CODER	47	1 Replace the cable. 2 Correct the cable layout (vicinity of the power line).	The A/B phase signal of the spindle position coder (connector JYA3) is abnormal. The relationship between the A/B phase and 1-rotation signal is incorrect (Pulse count mismatch).
SP9049	DEF. SPEED IS OVER VALUE	49	Check whether the calculated differential speed value exceeds the maximum motor speed.	In differential speed mode, the speed of the other spindle converted to the speed of the local spindle has exceeded the allowable limit (the differential speed is calculated by multiplying the speed of the other spindle by the gear ratio).
SP9050	SYNCRONOUS VALUE IS OVER SPEED	50	Check whether the calculated value exceeds the maximum motor speed.	In spindle synchronization, the speed command calculation value exceeded the allowable limit (the motor speed is calculated by multiplying the specified spindle speed by the gear ratio).
SP9051	PS LOW VOLT. DC LINK		1 Check and correct the power supply voltage. 2 Replace the MC.	Input voltage drop was detected. (Power Supply alarm (PS) indication: 04) (Momentary power failure or poor MCC contact)
SP9052	ITP FAULT 1	52	1 Replace the Spindle Amplifier (SP) control printed circuit board. 2 Replace the main board or additional spindle board in the CNC.	An abnormality is detected in the interface between the CNC and spindle amplifier (the ITP signal stopped).
SP9053	ITP FAULT 2	53	1 Replace the Spindle Amplifier (SP) control printed circuit board. 2 Replace the main board or additional spindle board in the CNC.	An abnormality is detected in the interface between the CNC and spindle amplifier (the ITP signal stopped).
SP9054	OVERCURRENT	54	Review the load state.	An overload current was detected.
SP9055	ILLEGAL POWER LINE	55	1 Replace the magnetic contactor. 2 Check and correct the sequence.	The power line state signal of the magnetic contactor for spindle switching/speed range switching is abnormal.

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9056	COOLING FAN FAILURE	56	Replace the internal cooling fan.	The internal cooling fan stopped.
SP9057	PS EXCESS-REGENERATION2		1 Decrease the acceleration/deceleration duty. 2 Check the cooling condition (peripheral temperature). 3 If the cooling fan stops, replace the resistor. 4 If the resistance is abnormal, replace the resistor.	An overload was detected in the regenerative resistance. (Power Supply alarm indication: 16) Thermostat operation or short-time overload was detected. The regenerative resistor was disconnected, or an abnormal resistance was detected.
SP9058	PS OVERLOAD		1 Check the Power Supply (PS) cooling status. 2 Replace the Power Supply (PS).	The temperature of the radiator of the Power Supply (PS) has increased abnormally. (Power Supply (PS) alarm indication: 03)
SP9059	PS INTERNAL FAN FAILURE		Replace the Power Supply (PS).	The internal cooling fan for the Power Supply (PS) stopped. (Power Supply alarm indication: 02)
SP9061	SEMI-FULL ERROR EXCESS	61	Check parameter settings.	The error between the semi-closed and full-closed sides when the dual position feedback function is used is too large.
SP9065	SERIAL SPINDLE ALARM	65	1 Check parameter settings. 2 Check sensor connections and signals. 3 Check power line connections.	The move distance is too long when the magnetic pole is confirmed. (synchronous spindle motor)
SP9066	COM. ERROR BETWEEN SP AMPS	66	1 Replace the cable. 2 Check and correct the connection.	An error was found in communication (connector JX4) between Spindle Amplifiers (SP).
SP9067	FSC/EGB COMMAND ERROR	67	Check the sequence (reference position return command).	In the spindle EGB mode, reference position return was specified.
SP9069	SAFETY SPEED OVER	69	1 Check the speed command. 2 Check parameter settings. 3 Check the sequence.	In the state in which safety speed monitoring was enabled, the system detected that the motor speed exceeded the safety speed or detected an error during a free-run stop.

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9070	ILLEGAL AXIS DATA	70	1 When this alarm is issued because the configuration of the spindle amplifiers (SPs) was changed, set the spindle number for the spindle amplifier (bit 7 of parameter No. 4541 to 1 and then 0, and turn the power to the entire system off). 2 Replace the spindle amplifier.	An error was detected during transfer of spindle data of a spindle.
SP9071	SAFETY PARAMETER ERROR	71	1 Input the safety parameter again. 2 Replace the Spindle Amplifier (SP) control printed-circuit board.	An error was detected in an axis parameter check.
SP9072	MISMATCH RESULT OF MOTOR SPEED CHECK	72	1 Replace the Spindle Amplifier (SP) control printed-circuit board. 2 Replace the main board or additional spindle board in the CNC.	A mismatch was detected between the safety speed check results of the Spindle Amplifier (SP) and those of the CNC.
SP9073	MOTOR SENSOR DISCONNECTED	73	1 Replace the feedback cable. 2 Check the shield. 3 Check and correct the connection. 4 Adjust the sensor.	The motor sensor feedback signal is not present. (connector JYA2)
SP9074	CPU TEST ERROR	74	Replace the Spindle Amplifier (SP) control printed-circuit board.	An error was detected in a CPU test.
SP9075	CRC ERROR	75	Replace the Spindle Amplifier (SP) control printed circuit board.	An error was detected in a ROM CRC test.
SP9076	INEXECUTION OF SAFETY FUNCTIONS	76	Replace the Spindle Amplifier (SP) control printed-circuit board.	The spindle amplifier (SP) detected that the safety function was not executed.
SP9077	MISMATCH RESULT OF AXIS NUMBER CHECK	77	1 Replace the Spindle Amplifier (SP) control printed-circuit board. 2 Replace the main board or additional spindle board in the CNC.	A mismatch was detected between the axis number check results of the Spindle Amplifier (SP) and those of the CNC.
SP9078	MISMATCH RESULT OF SAFETY PARAMETER CHECK	78	1 Replace the Spindle Amplifier (SP) control printed-circuit board. 2 Replace the main board or additional spindle board in the CNC.	The result of a safety parameter check by the spindle amplifier (SP) conflicts with that of a safety parameter check by the CNC.
SP9079	INITIAL TEST ERROR	79	Replace the Spindle Amplifier (SP) control printed-circuit board.	An error was detected in a initial test operation.

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9080	ALARM AT THE OTHER SP AMP.	80	Remove the cause of the alarm of the remote Spindle Amplifier (SP).	During inter-Spindle Amplifier (SP) communication, an alarm was generated on the remote Spindle Amplifier (SP).
SP9081	1-ROT MOTOR SENSOR ERROR	81	1 Check and correct the parameter. 2 Replace the feedback cable. 3 Adjust the sensor.	The one-rotation signal of the motor sensor cannot be correctly detected. (connector JYA2)
SP9082	NO 1-ROT MOTOR SENSOR	82	1 Replace the feedback cable. 2 Adjust the sensor.	The one-rotation signal of the motor sensor is not generated. (connector JYA2)
SP9083	MOTOR SENSOR SIGNAL ERROR	83	1 Replace the feedback cable. 2 Adjust the sensor.	An irregularity was detected in a motor sensor feedback signal. (connector JYA2)
SP9084	SPNDL SENSOR DISCONNECTED	84	1 Replace the feedback cable. 2 Check the shield. 3 Check and correct the connection. 4 Check and correct the parameter. 5 Adjust the sensor.	The spindle sensor feedback signal is not present. (connector JYA4)
SP9085	1-ROT SPNDL SENSOR ERROR	85	1 Check and correct the parameter. 2 Replace the feedback cable. 3 Adjust the sensor.	The one-rotation signal of the spindle sensor cannot be correctly detected. (connector JYA4)
SP9086	NO 1-ROT SPNDL SENSOR	86	1 Replace the feedback cable. 2 Adjust the sensor.	The one-rotation signal of the spindle sensor is not generated. (connector JYA4)
SP9087	SPNDL SENSOR SIGNAL ERROR	87	1 Replace the feedback cable. 2 Adjust the sensor.	An irregularity was detected in a spindle sensor feedback signal. (connector JYA4)
SP9088	COOLING RADI FAN FAILURE	88	Replace the Spindle Amplifier radiator cooling fan.	The radiator cooling fan stopped.
SP9089	SUB MODULE SM (SSM) ERROR	89	1 Check the connection between the Spindle Amplifier (SP) and the submodule SM (SSM). 2 Replace the submodule SM(SSM). 3 Replace the Spindle Amplifier (SP) control printed-circuit board.	Submodule SM (SSM) error (synchronous spindle motor)
SP9090	UNEXPECTED ROTATION	90	1 Check magnetic pole detection operation. 2 Check whether the rotor and sensor are aligned correctly.	Unexpected rotation of the synchronous spindle motor was detected.
SP9091	POLE POSITION COUNT MISS	91	Replace the motor sensor cable.	Count error of the magnetic pole position of the synchronous spindle motor

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9092	OVER SPEED TO VELOCITY COMMAND	92	Check the sequence (whether SFR or SRV is turned on and off in the position control mode).	The motor speed exceeds the overspeed level corresponding to the velocity command.
SP9110	AMP COMMUNICATION ERROR	b0	1 Replace the communication cable between Spindle Amplifier (SP) and Power Supply (PS). 2 Replace the Spindle Amplifier (SP) or Power Supply (PS) control printed circuit board.	Communication error between Spindle Amplifier (SP) and Power Supply (PS)
SP9111	PS LOW VOLT. CONTROL		Replace the Power Supply (PS) control printed circuit board.	Low converter control power supply voltage (Power Supply indication : 06)
SP9112	PS EXCESS-REGENERATION1		1 Check the regenerative resistance. 2 Check the motor selection. 3 Replace the Power Supply (PS).	Excessive converter regenerative power (Power Supply indication : 08)
SP9113	PS EXTERNAL FAN FAILURE		Replace the external radiator cooling fan for Power Supply (PS).	Stopped the external radiator cooling fan for Power Supply (PS) (Power Supply indication = 10)
SP9114	PS CONTROL AXIS ERROR 1	b4	Set parameter No. 4657 to 0. Or set parameter APS (No.11549#0) to 1 and execute automatic setting.	Invalid parameter
SP9115	PS CONTROL AXIS ERROR 2	b5	Set parameter No. 4657 to other than 0. Or set parameter APS (No.11549#0) to 1 and execute automatic setting.	Invalid parameter
SP9120	COMMUNICATION DATA ERROR	C0	1 Replace the communication cable between CNC and Spindle Amplifier (SP). 2 Replace the Spindle Amplifier (SP) control printed circuit board. 3 Replace the main board or additional spindle board in the CNC.	Communication data alarm
SP9121	COMMUNICATION DATA ERROR	C1	1 Replace the communication cable between CNC and Spindle Amplifier (SP). 2 Replace the Spindle Amplifier (SP) control printed circuit board. 3 Replace the main board or additional spindle board in the CNC.	Communication data alarm

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9122	COMMUNICATIO N DATA ERROR	C2	1 Replace the communication cable between CNC and Spindle Amplifier (SP). 2 Replace the Spindle Amplifier (SP) control printed circuit board. 3 Replace the main board or additional spindle board in the CNC.	Communication data alarm
SP9123	SPINDLE SWITCH CIRCUIT ERROR	C3	Replace the submodule SW(SSW).	Submodule SW (SSW) error (spindle switching)
SP9124	LEARNING CTRL ROTATION CMD ERROR	C4	1 Correct the velocity command. 2 Correct the setting of the learning period parameter (No. 4425).	A velocity that could not be used for learning control was specified in time-based learning control.
SP9125	LEARNING CTRL COMP ORDER ERROR	C5	Correct the settings of the parameters for the maximum and minimum orders of dynamic characteristic compensation elements for learning control (Nos. 4427 and 4428).	An order outside the allowable range was specified for dynamic characteristic compensation elements.
SP9127	LEARNING CTRL PERIOD ERROR	C7	Correct the setting of the parameter for the number of divisions per period for learning control (No. 4425).	In angle-based learning control, the reference angle period is invalid.
SP9128	SP SYNC VELOCITY ERROR EXCESS	C8	1 Check the sequence (whether SFR or SRV is turned off or on in position control). 2 Check and correct the load status. 3 Check the parameter setting.	In spindle synchronous control, velocity error exceeds the setting.
SP9129	SP SYNC POSITION ERROR EXCESS	C9	1 Check the sequence (whether SFR or SRV is turned off or on in position control). 2 Check and correct the load status. 3 Check the parameter setting.	In spindle synchronous control, position error exceeds the setting.
SP9130	TORQUE TANDEM POLARITY ERROR	d0	Check the setting of the parameter for the rotation direction (bit 2 of No. 4353).	In tandem control, the rotation directions of the master and slave motors are invalid.
SP9131	SPINDLE TUNING FUNCTION ALARM	d1	Check the message displayed by SERVO GUIDE.	Spindle tuning function alarm

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9132	SER.SENSOR DATA ERROR	d2	1 Replace the feedback cable. 2 Replace the sensor. 3 Replace the Spindle Amplifier (SP).	Serial data error between a serial sensor and spindle amplifier (SP)
SP9133	SER.SENSOR TRANSFER ERROR	d3	1 Replace the feedback cable. 2 Replace the sensor. 3 Replace the Spindle Amplifier (SP).	Serial data error between a serial sensor and spindle amplifier (SP)
SP9134	SER.SENSOR SOFT PHASE ERROR	d4	1 Check and correct the sensor parameter setting. 2 Take action against noise. 3 Replace the sensor.	The change in serial sensor positional data is too large.
SP9135	SAFETY SPEED ZERO ERROR(SP)	d5	Perform operation within the safety speed zero range.	The motor position exceeded the safety speed zero monitoring width.
SP9136	MISMATCH RESULT OF SAFETY SPEED ZERO CHECK(SP)	d6	Replace the Spindle Amplifier (SP) control printed circuit board.	The Spindle Amplifier (SP) speed zero determination result did not match the CNC speed zero determination result.
SP9137	SP DEVICE COMMUNICATIO N ERROR	d7	Replace the Spindle Amplifier (SP) control printed circuit board.	Device communication error in the spindle amplifier (SP)
SP9138	CURRENT LIMIT SETTING ERROR	d8	Check the parameter.	The current limit level setting is out of the specified range.
SP9139	SER.SENSOR PULSE MISS	d9	Replace the sensor.	An error occurred in the serial sensor interpolation circuit.
SP9140	SER.SENSOR COUNT MISS	E0	1 Take action against noise. 2 Replace the sensor.	The number of feedback pulses per one-rotation signal of the serial sensor is outside the specified range.
SP9141	SER.SENSOR NO 1-ROT SIGNAL	E1	1 Check and correct the sensor parameter setting. 2 Replace the sensor.	The serial sensor is not placed in the status in which the one-rotation signal has been detected.
SP9142	SER.SENSOR ABNORMAL	E2	Replace the sensor.	A serial sensor error occurred.
SP9143	CS HIGH SPEED CHANGE CMD ERROR	E3	Check the sequence.	A command was issued though the one-rotation signal had not been detected.
SP9144	CURRENT DETECT CIRCUIT ERROR	E4	Replace the Spindle Amplifier (SP).	An error was found in the current detection circuit.
SP9145	LOW VOLTAGE DRIVER	E5	Replace the Spindle Amplifier (SP).	The driver circuit voltage has dropped.

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9146	SP:INTERNAL OVERHEAT	E6	1 Check that the temperature in the power magnetics cabinet is within the specification range. 2 If this alarm is issued immediately after power-on, replace the spindle amplifier (SP).	The internal temperature of the spindle amplifier (SP) exceeds the specified value.
SP9147	SP:GROUND FAULT	E7	Replace the motor or amplifier.	A ground fault occurred in the motor or amplifier.
SP9148	AXIS NUMBER NOT SET	E8	Turn the power to the entire system off.	The spindle number of the spindle (SP) is not set.
SP9149	EXT.CURRENT FB U-OFFSET	E9	Replace the Spindle Amplifier (SP) control printed circuit board.	An error was detected in the part of spindle amplifier (SP).
SP9150	EXT.CURRENT FB V-OFFSET	F0	Replace the Spindle Amplifier (SP) control printed circuit board.	An error was detected in the part of spindle amplifier (SP).
SP9151	FILTER MODULE ERROR	F1	Replace the filter module.	A filter module error was detected.
SP9152	EXT.CUR.FB DISCONNECT	F2	Replace the cable (JYA4).	Disconnection of the external current sensor was detected.
SP9153	SP NO FAILURE	F3	Check the parameter.	No hardware error is determined by a self check.
SP9154	PHASE OPEN	F4	Replace the motor.	A motor error was detected.
SP9155	FAILURE OF SP (OPEN)	F5	Replace the Spindle Amplifier (SP).	An error was detected in the spindle amplifier (SP).
SP9156	FAILURE OF CURRENT CTRL.	F6	Replace the Spindle Amplifier (SP) control printed circuit board.	An error was detected in the part of spindle amplifier (SP).
SP9157	FAILURE OF SP (SHORT)	F7	Replace the Spindle Amplifier (SP).	An error was detected in the spindle amplifier (SP).
SP9159	MISMATCHED FUNCTION CODE	F9	Turn the power off, then restart.	CNC, SV, SP or PS software has been update.
SP9200	PS GROUND FAULT		Replace the motor or amplifier.	A ground fault occurred in the motor or amplifier. (Power Supply (PS) alarm indication: 09)
SP9201	PS OVERCURRENT 2		Replace the Power Supply (PS).	Overcurrent was detected in the main circuit input in the power supply (PS). (Power Supply (PS) alarm indication: 11)
SP9202	PS OVERCURRENT 3		Replace the Power Supply (PS).	Overcurrent was detected in the main circuit input in the power supply (PS). (Power Supply (PS) alarm indication: 12)
SP9203	PS OVERCURRENT 4		Replace the Power Supply (PS).	Overcurrent was detected in the main circuit input in the power supply (PS). (Power Supply (PS) alarm indication: 13)

Number	Message	SP indication (*1)	Faulty location and remedy	Description
SP9204	PS SOFT THERMAL		Review the current condition.	Overload was detected in the power supply (PS). (Power Supply (PS) alarm indication: 15)
SP9205	PS OVER VOLT. DC LINK 2		Replace the Power Supply (PS).	Overvoltage was detected in the DC link section. (Power Supply (PS) alarm indication: 17)
SP9206	PS OVER VOLT. DC LINK 3		Replace the Power Supply (PS).	Overvoltage was detected in the DC link section. (Power Supply (PS) alarm indication: 18)
SP9207	PS OVER VOLT. DC LINK 4		Replace the Power Supply (PS).	Overvoltage was detected in the DC link section. (Power Supply (PS) alarm indication: 19)
SP9208	PS LOW VOLT.DC LINK2		Replace the Power Supply (PS).	Voltage drop was detected in the DC link section. (Power Supply (PS) alarm indication: 20)
SP9209	PS LOW VOLT.DC LINK3		Replace the Power Supply (PS).	Voltage drop was detected in the DC link section. (Common Power Supply (PS) alarm indication: 21)
SP9210	PS LOW VOLT.DC LINK4		Replace the Power Supply (PS).	Voltage drop was detected in the DC link section. (Power Supply (PS) alarm indication: 22)
SP9211	PS ILLEGAL PARAMETER		Change the parameter to an appropriate value.	An invalid value was set in a parameter for controlling the power supply (PS). (Power Supply (PS) alarm indication: 23)
SP9212	PS HARDWARE ERROR		Replace the Power Supply (PS).	A hardware error was detected in the power supply (PS). (Power Supply (PS) alarm indication: 24)
SP9213	PS SUB MODULE ERROR 1		Replace Power Supply SUB MODULE (PS SUB MODULE).	A error was detected in the Power Supply SUB MODULE (PS SUB MODULE). (Power Supply alarm indication: 25)
SP9214	PS SUB MODULE ERROR 2		Replace Power Supply SUB MODULE (PS SUB MODULE).	A error was detected in the Power Supply SUB MODULE (PS SUB MODULE). (Power Supply alarm indication: 26)
SP9215	PS SUB MODULE ERROR 3		Replace Power Supply SUB MODULE (PS SUB MODULE).	A error was detected in the Power Supply SUB MODULE (PS SUB MODULE). (Power Supply alarm indication: 27)
SP9216	PS SUB MODULE ERROR 4		Replace Power Supply SUB MODULE (PS SUB MODULE).	A error was detected in the Power Supply SUB MODULE (PS SUB MODULE). (Power Supply alarm indication: 28)

Error codes (serial spindle)

NOTE

*1 Note that the meanings of the Spindle Amplifier (SP) indications differ depending on which LED, the red or yellow LED, is on. When the yellow LED is on, an error code is indicated with a 2-digit number. An error code is indicated in the CNC diagnosis data No.710. When the red LED is on, the Spindle Amplifier (SP) indicates the number of an alarm generated in the serial spindle.
→ See Item, “Alarm list (serial spindle).”

Spindle amplifier indication (*1)	Description	Faulty section and action
01	Although neither *ESP (emergency stop signal; there are two types of signals including the input signal and Power Supply (PS) contact signal) nor MRDY (machine ready signal) is input, SFR (forward rotation signal)/SRF (reverse rotation signal)/ORCM (orientation command) is input.	Check the *ESP and MRDY sequence. For MRDY, pay attention to the parameter setting regarding the use of the MRDY signal (bit 0 of parameter No. 4001).
03	The parameter settings are such that a position sensor is not used (position control not performed) (bits 3, 2, 1, 0 of parameter No. 4002 = 0, 0, 0, 0), but a Cs contour control command is input. In this case, the motor is not excited.	Check the parameter settings.
04	The parameter settings are such that a position sensor is not used (position control not performed) (bits 3, 2, 1, 0 of parameter No.4002 = 0, 0, 0, 0), but a servo mode (rigid tapping, spindle positioning, etc.) or spindle synchronization command is input. In this case, the motor is not excited.	Check the parameter settings.
05	The orientation function option parameter is not specified, but ORCM (orientation command) is input.	Check the orientation function parameter settings.
06	The speed range switching control function option parameter is not specified, but low-speed characteristic winding is selected (RCH = 1).	Check the speed range switching control function parameter settings and the power line state check signal (RCH).
07	A Cs contour control command is input, but SFR (forward rotation command)/SRV (reverse rotation command) is not input.	Check the sequence.
08	A servo mode (rigid tapping, spindle positioning, etc.) control command is input, but SFR (forward rotation command)/SRV (reverse rotation command) is not input.	Check the sequence.
09	A spindle synchronization command is input, but SFR (forward rotation command)/SRV (reverse rotation command) is not input.	Check the sequence.
10	A Cs contour control command is input, but another mode (servo mode, spindle synchronization or orientation) is specified.	Do not switch to another mode during a Cs contour control command. Before moving to another mode, cancel the Cs contour control command.
11	A servo mode (rigid tapping, spindle positioning, etc.) command is input, but another mode (Cs contour control, spindle synchronization or orientation) is specified.	Do not switch to another mode during a servo mode command. Before moving to another mode, cancel the servo mode command.

Spindle amplifier indication (*1)	Description	Faulty section and action
12	A spindle synchronization command is input, but another mode (Cs contour control, servo mode or orientation) is specified.	Do not switch to another mode during a spindle synchronization command. Before moving to another mode, cancel the spindle synchronization command.
13	An orientation command is input, but another mode (Cs contour control, servo mode or spindle synchronization control) is specified.	Do not switch to another mode during an orientation command. Before moving to another mode, cancel the orientation command.
14	Both SFR (forward rotation command) and SRV (reverse rotation command) are input at the same time.	Issue either of them.
16	The parameter settings are such that the differential speed control function is not used (bit 5 of parameter No. 4000 = 0), but DEFMD (differential speed mode command) is input.	Check the parameter settings and differential speed mode command.
17	The speed detector parameter settings (bits 2, 1, and 0 of parameter No. 4011) are not valid. There is no corresponding speed detector.	Check the parameter settings.
18	The parameter settings are such that a position sensor is not used (position control not performed (bits 3, 2, 1, and 0 of parameter No. 4002), but position coder system orientation is issued.	Check the parameter settings and the input signal.
19	The magnetic sensor orientation command is input, but another mode (Cs contour control, servo mode or spindle synchronization control) is specified.	Do not switch to another mode during an orientation command. Before moving to another mode, cancel the orientation command.
21	The tandem operation command was input in the spindle synchronization control enable state.	Input the tandem operation command when spindle synchronization control is canceled.
22	Spindle synchronization control was specified in the tandem operation enable state.	Specify spindle synchronization control when torque tandem operation is canceled.
23	The tandem operation command is input without the required option.	Torque tandem control requires a CNC software option. Check the option.
24	If index is performed continuously in position coder method orientation, an incremental operation is performed first (INCMD = 1), then an absolute position command (INCMD = 0) is input.	Check INCMD (incremental command). If an absolute position command is to follow, be sure to perform absolute position command orientation first.
26	The parameter settings are such that both spindle switch and three-stage speed range switch are used.	Check the parameter settings and the input signal.
29	The parameter settings are such that the shortest-time orientation function is used (bit 6 of parameter No. 4018 = 0, Nos. 4320 to 4323 ≠0).	In the α*i* series spindle amplifier, the shortest-time orientation function cannot be used. The use of the optimum orientation function is recommended.
30	The magnetic pole has not been detected, but a command is input.	In the magnetic pole undetected state (EPFIXA = 0), the motor cannot be driven even when a command is input. Input a command in the magnetic pole detected state (EPFIXA = 1). When EPFSTR is set to 1, any command is ignored and this error is displayed even in the magnetic pole detected state. After the completion of magnetic pole detection, set EPFSTR to 0.
31	The hardware configuration is such that the spindle FAD function cannot be used. In this case, the motor is not activated.	Check the CNC model. With the FS30*i* series, the spindle FAD function is not used.

Spindle amplifier indication (*1)	Description	Faulty section and action
32	S0 is not specified in the velocity mode, but the disturbance input function is enabled (bit 7 of parameter No. 4395 is set to 1).	Specify S0 in the velocity mode before enabling the disturbance input function (bit 7 of parameter No. 4395 to 1).
33	The hardware configuration is such that the spindle EGB function cannot be used. In this case, the motor is not activated.	Check the CNC model.
34	Both the spindle FAD function and the spindle EGB function are enabled. In this case, the motor is not activated.	The two functions cannot be used at the same time. Enable either function only.
35	Spindle Amplifier (SP) ID information cannot be obtained.	Replace the spindle amplifier with one with correct ID information.
36	The submodule SM (SSM) is faulty .	For action to be taken, refer to the FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN).
37	The current loop setting (No. 4012) has been changed.	Check the setting of parameter No. 4012, and turn the power off, then on again.
38	A parameter related to communication between spindle amplifiers is specified incorrectly. Alternatively, a function unavailable with the torque tandem function is set.	Check the parameters.
39	Although SFR (forward rotation command), SRV (reverse rotation command), or ORCM (orientation command) is input, DSCN (disconnection detection disable signal) is input.	Check the sequence. Do not input DSCN (disconnection detection disable signal) during the input of a command which excites the motor.
43	A setting which does not support the αiCZ sensor (serial) is used.	Check the parameter settings.
44	The spindle amplifier does not support the control period setting.	Check the setting of parameter No. 4012.

(11) Overheat alarms (OH alarm)

Number	Message	Description
OH0700	LOCKER OVERHEAT	CNC cabinet overheat
OH0701	FAN MOTOR STOP	PCB cooling fan motor abnormality
OH0704	OVERHEAT	Spindle overheat due to detection of changes in the spindle speed - When the cutting load is large, offload the cutting conditions. - Check if the cutting tool became dull. - Check if the spindle amplifier malfunctions.

(12) Other alarms (DS alarm)

Number	Message	Description
DS0001	SYNC EXCESS ERROR (POS DEV)	In feed axis synchronization control, the difference in the amount of positional deviation between the master and slave axes exceeded the parameter No. 8323 setting value. This alarm occurs for the master or slave axis.
DS0002	SYNC EXCESS ERROR ALARM 1	In feed axis synchronization control, the difference in the amount of synchronization between the master and slave axes exceeded the parameter No. 8331 setting value. This alarm occurs only for the slave axis.
DS0003	SYNCHRONIZE ADJUST MODE	The system is in the synchronize adjust mode.
DS0004	EXCESS MAXIMUM FEEDRATE	The malfunction prevention function detected the command in which a value exceeding the maximum speed was specified.

Number	Message	Description
DS0005	EXCESS MAXIMUM ACCELERATION	The malfunction prevention function detected the command in which a value exceeding the maximum acceleration was specified.
DS0006	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0007	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0008	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0009	ILLEGAL EXECUTION SEQUENCE	The malfunction prevention function detected an illegal execution sequence.
DS0010	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an invalid reference area.
DS0011	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an invalid reference area.
DS0012	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an invalid reference area.
DS0013	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an invalid reference area.
DS0014	TOOL CHANGE DETECT MACHINE LOCK	A machine lock is turned on for the Z axis for which the tool is being changed.
DS0015	TOOL CHANGE DETECT MIRROR IMAGE	A mirror image is turned on for the Z axis for which the tool is being changed.
DS0016	SERIAL DCL:FOLLOW-UP ERROR	(1) The settings of parameters Nos.1883 and 1884 fall outside the range. (2) The current position at establishment of the origin subtracted by the distance between the reference positions (detection unit) exceeded ±2147483647. Change the current position or reference position to prevent this situation.
DS0017	SERIAL DCL:REF-POS ESTABLISH ERR	The travel amount at the FL speed at establishment of the origin exceeded the setting of parameter No. 14010.
DS0018	SERIAL DCL:MISMATCH(SSYNC CTRL)	Of the master and slave axes for feed axis synchronous control, one axis is a linear scale with the origin and the other is not a linear scale with the origin. In such a configuration, the feed axis synchronous control selection signal (SYNC<Gn138> or SYNCJ <Gn140>) needs to be set to 0 to establish the origin.
DS0019	SERIAL DCL:MISMATCH(ANGL-AXIS)	In angular axis control, one of the angular axis and the Cartesian axis is a linear scale with an origin, while the other is not a linear scale with an origin. Angular axis control cannot be used in such a configuration.
DS0020	REFERENCE RETURN INCOMPLETE	An attempt was made to perform an automatic return to the reference position on the perpendicular axis before the completion of a return to the reference position on the angular axis. However, this attempt failed because a manual return to the reference position during angular axis control or an automatic return to the reference position after power-up was not commanded. First, return to the reference position on the angular axis, then return to the reference position on the perpendicular axis.
DS0022	DUAL CHECK SAFETY IS NOT WORKED	Bit 6 (DCE) of parameter No. 1902 has set the Dual Check Safety function to be disabled.
DS0023	ILLEGAL PARAMETER (I-COMP VAL)	The setting of the inclination compensation parameter is incorrect. The compensation per compensation point is too large or too small.

Number	Message	Description
DS0024	UINT SIGNAL WAS ILLEGALLY INPUT	An interruption type custom macro was called during movement to the program restart position.
DS0025	G60 CANNOT BE EXECUTED	The state of a mirror image is different between the time when look-ahead of a block for unidirectional positioning was performed and the time when execution of the block was started, so unidirectional positioning cannot be performed. Modify the program.
DS0026	MISMATCH OF ANGULAR AXIS(D.C.S)	On angular axis control, one of the angular/perpendicular axes is the scale with ref-pos, and the other of them is not the scale with ref-pos. Such system is not admired.
DS0027	MISMATCH OF SYNCHRONOUS AXIS(D.C.S)	Master/slave axes of axis synchronous control, one of them is the linear scale with distance-coded reference marks, and the other of them is not the linear scale with distance-coded reference marks. Please establish reference position with the input signal SYNCn<G138>, SYNCJn<G140> or parameter setting to 0.
DS0029	UNAVAILABLE ROT AXIS MOVE IN TPC	In tool posture control, the rotation axis fell outside the set operation range (parameters Nos. 19741 to 19744). Review the machine configuration and command.
DS0030	TOOL DIRC CMP IMPOSSIBLE	When tool center point control is performed in the angular surface machining command mode or workpiece placement error compensation is performed in a 5-axis cutting machine, if settings are made to make compensation in the tool direction (bit 0 (RCM) of parameter No. 11200 is 1), the position of the rotation axis could not be obtained.
DS0050	TOO MANY SIMULTANEOUS AXES	A movement was performed along more axes than can be controlled by simultaneous axis control. Check whether a command in the program is specified for more axes than can be controlled by simultaneous axis control.
DS0059	SPECIFIED NUMBER NOT FOUND	[External data I/O] The No. specified for a program No. or sequence No. search could not be found. There was an I/O request issued for a pot No. or offset (tool data), but either no tool numbers have been input since power ON or there is no data for the entered tool No. [External workpiece No. search] The program corresponding to the specified workpiece No. could not be found.
DS0070	SUPERIMPOSE FOR HIGH-SPEED CYCLE CANNOT BE USED	Superimposed control for high-speed cycle machining is not enabled. Alternatively, the status is not the advanced superimposition state.
DS0071	START OR RELEASE CANNOT BE DONE	1) To start or cancel the inter-path flexible synchronous mode, the tool must be stopped along all axes. 2) To start or cancel advanced superimposition, movement along axes must stop.
DS0072	MANUAL REFERENCE RETURN CANNOT BE DONE	Manual reference position return cannot be performed in the advanced superimposition state.
DS0131	TOO MANY MESSAGE	An attempt was made to display an external operator message or external alarm message, but five or more displays were required simultaneously.
DS0132	MESSAGE NUMBER NOT FOUND	An attempt to cancel an external operator message or external alarm message failed because the specified message number was not found.
DS0133	TOO LARGE NUMBER	A value other than 0 to 4095 was specified as the external operator message or the external alarm message number.

Number	Message	Description
DS0300	APC ALARM: NEED REF RETURN	A setting to zero position for the absolute position detector (association with reference position and the counter value of the absolute position detector) is required. Perform the return to the reference position. This alarm may occur with other alarms simultaneously. In this case, other alarms must be handled first.
DS0306	APC ALARM: BATTERY VOLTAGE 0	The battery voltage of the absolute position detector has dropped to a level at which data can no longer be held. Or, the power was supplied to the Pulsecoder for the first time. The battery or cable is thought to be defective. Replace the battery with the machine turned on.
DS0307	APC ALARM: BATTERY LOW 1	The battery voltage of the absolute position detector has dropped to a level at which a replacement is required. Replace the battery with the machine turned on.
DS0308	APC ALARM: BATTERY LOW 2	The battery voltage of the absolute position detector dropped to a level at which a replacement was required in the past. (including during power off) Replace the battery with the machine turned on.
DS0309	APC ALARM: REF RETURN IMPOSSIBLE	An attempt was made to set the zero point for the absolute position detector by MDI operation when it was impossible to set the zero point. Rotate the motor manually at least one turn, and set the zero position of the absolute position detector after turning the CNC and servo amplifier off and then on again.
DS0310	NOT ON RETURN POINT	The return position recorded during retraction is not reached during recovery. The position may be displaced during recovery due to a machine lock or mirror image. Perform the operation again after making a reset.
DS0405	ZERO RETURN END NOT ON REF	The axis specified in automatic zero return was not at the correct zero point when positioning was completed. Perform zero return from a point whose distance from the zero return start position to the zero point is 2 or more revolutions of the motor. Other probable causes are: - The positional deviation after triggering the deceleration dog is less than 128. - Insufficient voltage or malfunctioning Pulsecoder.
DS0608	SV COOLING FAN FAILURE	Internal agitating fan failure.
DS0609	SV RADIATOR FAN FAILURE	Radiator cooling fan failure.
DS0610	PS INTERNAL FAN FAILURE	Internal agitating fan failure.
DS0611	PS EXTERNAL FAN FAILURE	External radiator cooling fan failure.
DS0612	PS OVERLOAD	Overheat
DS0613	PS IMPROPER INPUT POWER	Input power supply fault
DS0651	ACC.ERROR	Communication between the separate detector interface unit and acceleration sensor was disconnected.
DS1120	UNASSIGNED ADDRESS (HIGH)	The upper 4 bits (EIA4 to EIA7) of an external data I/O interface address signal are set to an undefined address (high bits).
DS1121	UNASSIGNED ADDRESS (LOW)	The lower 4 bits (EIA0 to EIA3) of an external data I/O interface address signal are set to an undefined address (low bits).
DS1124	OUTPUT REQUEST ERROR	OUTPUT REQUEST ERROR An output request was issued during external data output, or an output request was issued for an address that has no output data.
DS1128	DI.EIDLL OUT OF RANGE	The numerical value input by external data input signals ED0 to ED31 has exceeded the permissible range.

Number	Message	Description
DS1130	SEARCH REQUEST NOT ACCEPTED	No requests can be accepted for a program No. or a sequence No. search as the system is not in the memory mode or the reset state.
DS1131	EXT-DATA ERROR (OTHER)	[External Data I/O] An attempt was made to input tool data for tool offset by a tool No. during loading by the G10 code.
DS1150	A/D CONVERT ALARM	A/D converter malfunction
DS1184	PARAMETER ERROR IN TORQUE	An invalid parameter was set for torque control. The torque constant parameter is set to “0”.
DS1185	OVER MAXIMUM FEED	The maximum cutting feedrate or rapid traverse feedrate was exceeded in G54.3.
DS1448	ILLEGAL PARAMETER (D.C.S.)	The setting value of parameter for reference marks is satisfied the following any conditions. - The absolute-position detection function is enabled. - Either parameter 1821 (mark-1 interval) or parameter 1882 (mark-2 interval) is set to 0. - Parameters 1821 and 1882 have identical settings. - The difference between the settings made for parameters 1821 and 1882 is greater than or equal to twice either setting. - The setting value of parameters 1883 and 1884 are over the valid data range.
DS1449	REFERENCE MARK ARE DIFFERENT FROM PARAMETER	In case of distance coded linear scale I/F, the actual interval of reference marks is different from parameters Nos.1821 and 1882 setting value.
DS1450	ZERO RETURN NOT FINISHED	1st reference position return (CDxX7 to CDxX0: 17h (Hex)) was specified when the manual reference position return was not executed with the reference position return function enabled (bit 0 (ZRN) of parameter No. 1005 set to “0”).
DS1451	IMPROPER PMC AXIS COMMAND	The PMC axes cannot be controlled in this state.
DS1512	EXCESS VELOCITY	The feedrate of the linear axis during polar coordinate interpolation exceeded the maximum cutting feedrate.
DS1514	ILLEGAL MOTION IN G12.1 MODE	In a hypothetical axis direction compensation during the polar coordinate interpolation mode, an attempt is made to travel to the area in which the travel cannot be made.
DS1553	EXCESS VELOCITY IN G43.4/G43.5	The axis rate was attempt to exceed the maximum cutting feedrate and travel by the pivot tool length compensation.
DS1710	ILLEGAL ACC. PARAMETER (OPTIMUM TORQUE ACC/DEC)	There are errors in the parameters of permissible acceleration for Optimum Torque Acceleration/Deceleration. One of the following is the cause. 1) The ratio of the acceleration for deceleration to the acceleration for the acceleration is lower than the limited value. 2) The time to decelerate to 0 is larger than the maximum.
DS1711	ILLEGAL ACC. PARAMETER (RIGID TAPPING OPTIMUM ACC/DEC)	The permissible acceleration parameter for rigid tapping optimum acceleration/deceleration contains an error. The cause is one of the following: 1) The ratio of the deceleration to the acceleration is less than 1/3. 2) The time required to slow down to a speed of 0 exceeds the maximum. 3) The maximum acceleration (parameters Nos. 11421 to 11424) is 0.
DS1931	MACHINE PARAMETER INCORRECT	One of parameters Nos. 19665 to 19667 and Nos.19680 to 19744 used to configure the machine contains an error.

Number	Message	Description
DS1932	DI.THML SIGNAL ON	One of the parameters used to configure the machine is rewritten while the tool direction thermal displacement compensation function is enabled.
DS1933	NEED REF RETURN(SYNC:MIX:OVL)	The relation between a machine coordinate of an axis in synchronization, composition, or superposition control, and the absolute, or relative coordinate was displaced. Perform the manual return to the reference position.
DS2003	ILLEGAL USE FOR SERVO MOTOR SPINDLE	The servo axes for spindle use of the spindle control with servo motor was used by the following functions. - PMC axis control - Chopping
DS5340	PARAMETER CHECK SUM ERROR	Since a parameter was changed, the parameter checksum did not match the reference checksum. Set the original value to the parameter or set the reference checksum again.
DS5387	CAN NOT START REFERENCE RETURN WITH MECHANICAL STOPPER SETTING	The slave axis is not within the in-position width, for example, when the distance between the mechanical stoppers for the master and slave axes is larger than the withdrawal distance specified in parameter No. 7181 or 7182 for the master axis. Adjust the position of the mechanical stoppers or the setting of parameter No. 7181 or 7182.
DS5550	AXIS IMMEDIATE STOP	The movement along an axis was stopped immediately by the axis immediate stop function.

(13) Malfunction prevention function alarms (IE alarm)

Number	Message	Description
IE0001	+ OVERTRAVEL (SOFT 1)	The malfunction prevention function detected that stored stroke check 1 on the positive side was exceeded.
IE0002	- OVERTRAVEL (SOFT 1)	The malfunction prevention function detected that stored stroke check 1 on the negative side was exceeded.
IE0003	+ OVERTRAVEL (SOFT 2)	The malfunction prevention function detected that stored stroke check 2 on the positive side was exceeded.
IE0004	- OVERTRAVEL (SOFT 2)	The malfunction prevention function detected that stored stroke check 2 on the negative side was exceeded.
IE0005	+ OVERTRAVEL (SOFT 3)	The malfunction prevention function detected that stored stroke check 3 on the positive side was exceeded.
IE0006	- OVERTRAVEL (SOFT 3)	The malfunction prevention function detected that stored stroke check 3 on the negative side was exceeded.
IE0007	EXCESS MAXIMUM REV. DATA	The malfunction prevention function detected the command in which a value exceeding the maximum speed was specified.
IE0008	ILLEGAL ACC/DEC	The malfunction prevention function detected the acceleration/deceleration error.

I PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING

Appendix I, "PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING", consists of the following sections:

I.1 PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING 2266
I.2 NAMING RULES.....2275
I.3 RULES OF CHARACTERS IN PROGRAM FILE 2276
I.4 ERROR MESSAGE AND NOTE.....2277

I.1 PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING

Overview

By using this PC tool, you can make the memory card program file ("FANUCPRG.BIN") which is needed for the function "Memory Card Program Operation/Editing".
The maximum size of the memory card program file is 2048 Mbytes (2 Gbytes). The "Memory Card Program Operation/Editing" function needs the memory card which has the memory card program file on the FAT formatted memory card.
This PC tool can be operated on a PC in the marketplace and with following OS :

- Windows(R)NT4.0 Workstation (SP5 or later)
- Windows(R)2000 Professional (SP4 or later)
- Windows(R)XP Professional (SP2 or later)
- Windows(R)Vista Ultimate

And acceptable specification is as followings :

- Memory : 32Mbytes or more
- Hard disk : 10Mbytes or more free space and additional space for the memory card program file

I.1.1 Usage Notes

Before using this PC tool, please make sure there is no [temp] folder on the same place of this PC tool.
[temp] folder is created and used by this PC tool as work folder.
If [temp] folder is created, please do not access this folder.
[temp] folder and the files located in this folder will be deleted by this PC tool.

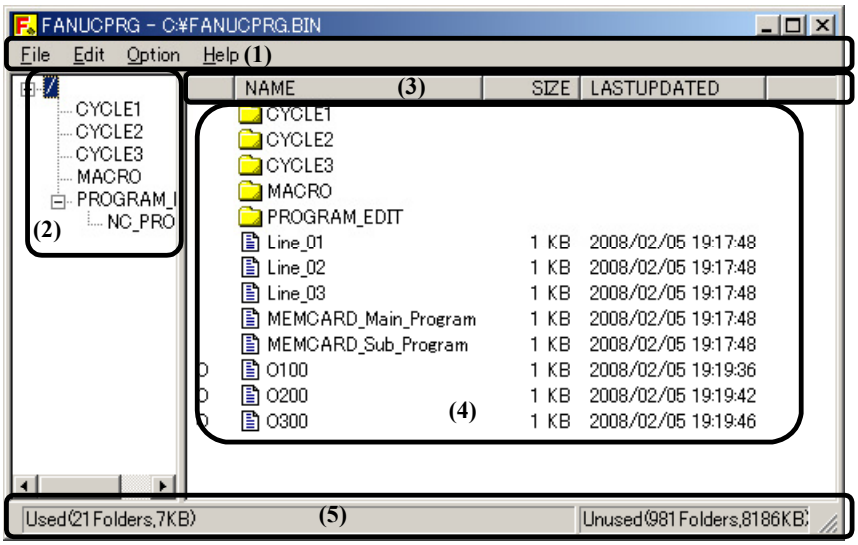
I.1.2 List of Functions of PC Tool

- Browsing the folders of the memory card program file
- Adding a program to the memory card program file by dropping the program into this PC tool from Explorer etc. (hereinafter referred to as "drop in")
- Extracting a program in the memory card program file as a text file on Windows file system by dropping out from this PC tool into Explorer etc. (hereinafter referred to as "drop out")
- Renaming a program in the memory card program file
- Deleting a program in the memory card program file
- Creating a new folder into the memory card program file
- Renaming a folder in the memory card program file
- Deleting a folder in the memory card program file
- Display of free space on the memory card program file

- Sorting list view of the memory card program file

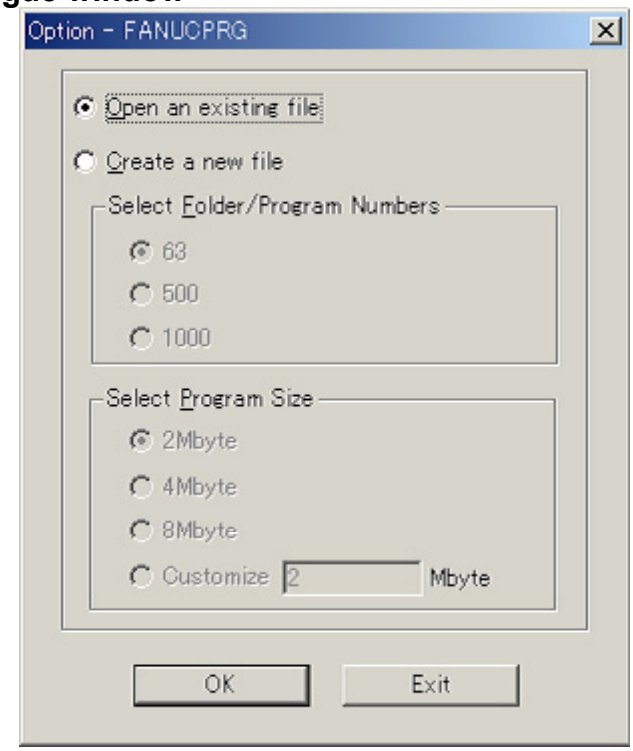
I.1.3 Explanation of Operations

- Outline of screen



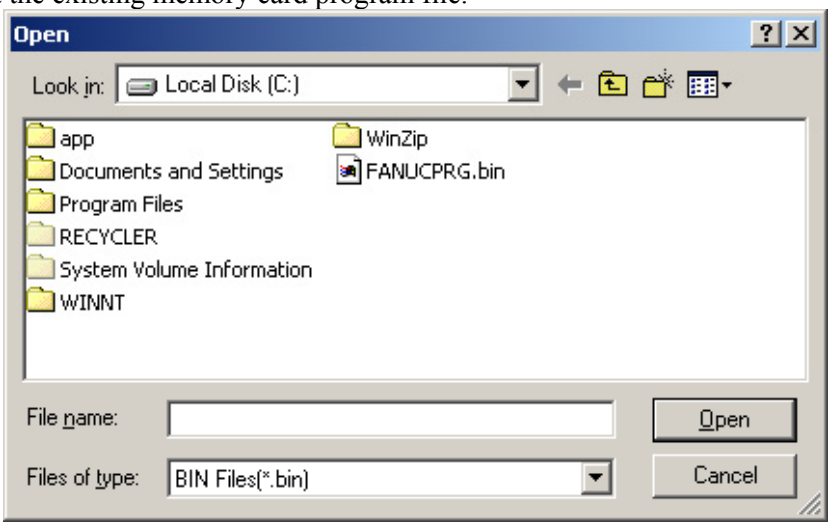
1) Menu bar : The menu of this PC tool is displayed.
2) Tree view : Browsing the folders of the memory card program file.
3) Column : Attributes of each file or folder in the memory card program file.
4) List view : Contents of selected folder are displayed.
5) Status bar : Used and Unused space in the memory card program file are displayed.

- Initial Option dialogue window

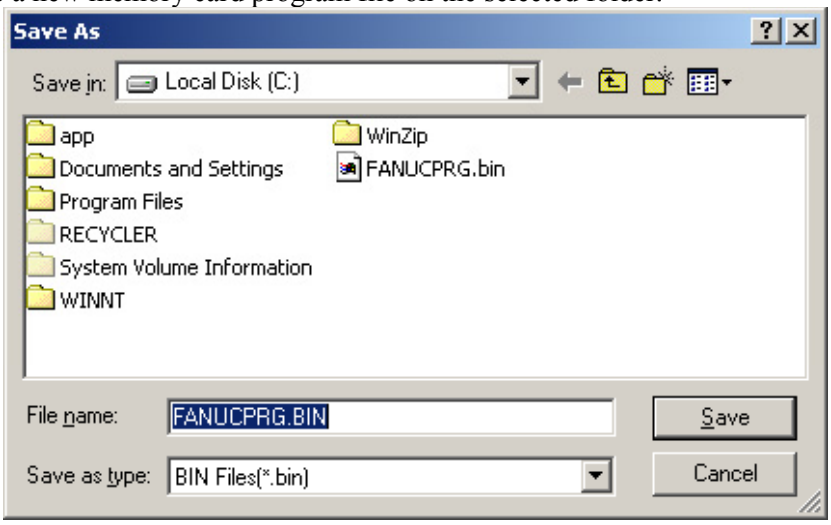


When this PC tool starts up, the Option dialogue window is displayed. Please select "Open an existing file" or "Create a new file".

- **When "Open an existing file" is selected**
  After OK button pushed, "Open" dialogue window is displayed.
  Please select the existing memory card program file.



- **When "Create a new file" is selected**
  After OK button pushed, "Save As" dialogue window is displayed.
  Please create a new memory card program file on the selected folder.



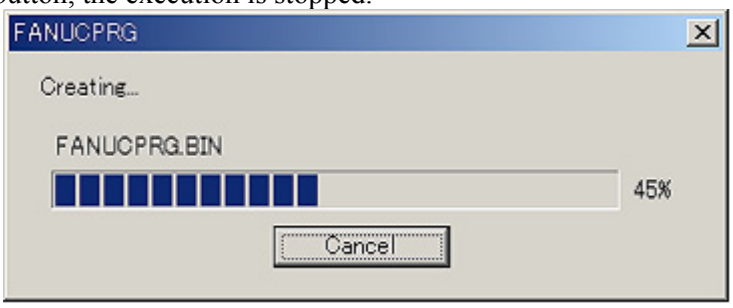
When the new memory card program file is created, the following items need to be selected:

- Folder/Program Numbers
- Program Size

"Folder/Program Numbers" can be selected among 63 / 500 / 1000. The default value is 63.
"Program Size" can be selected among 2Mbyte, 4Mbyte, 8Mbyte, and Customize. The default value is 2Mbyte.

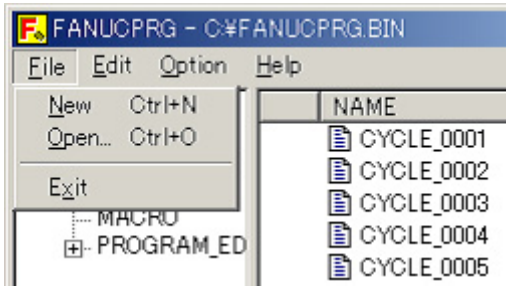
NOTE
1 When "Customize" size is selected, it is available to cover the range from 2Mbyte to 2048Mbyte.
2 Though the maximum size is 2048Mbyte, a slight loss exists by system use.
3 The number of "Folders" in the status bar includes number of program file and folder.

During creating of the memory card program file, the progress bar is being displayed. This progress bar is also displayed during executing of Drop-in and Drop-out.
If you push [Cancel] button, the execution is stopped.



- Menu

File menu

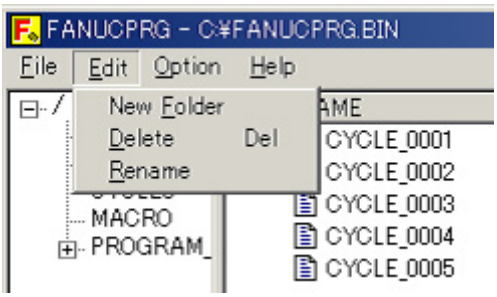


[New]
Create a new memory card program file.

[Open...]
Open the existing memory card program file.

[Exit]
Terminate this PC tool.

Edit menu

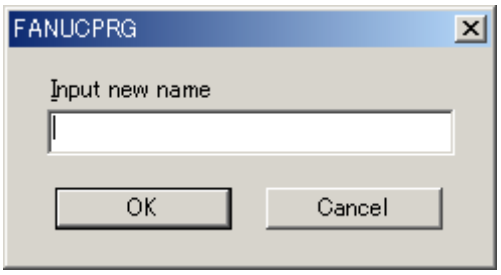


[New Folder]
Create new folder. It is available during Tree view selected.
Up to seven hierarchical levels starting from the user root folder (/USER) are permitted.
/USER/PATH1/Aaa/Bbb/Ccc/Ddd/O123
1 2 3 4 5 6 7(not folder)

[Delete]
Delete program files or folders.
If you delete a folder, all folders and program files in the folder will be deleted.

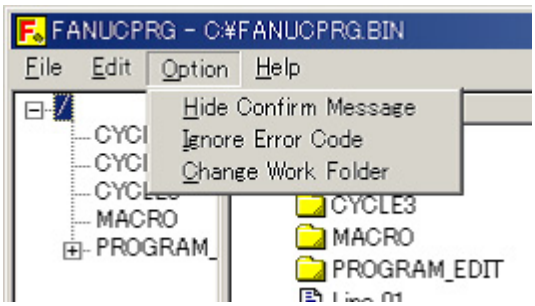
[Rename]

Rename a folder or file.



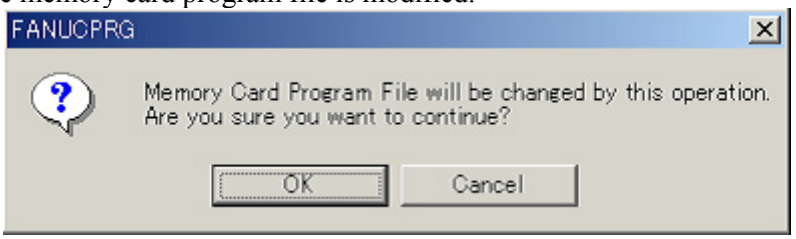
NOTE
For naming folder and program file, characters which can be used are limited. Please refer to "Naming rules".

Option menu



[Hide Confirm Message]

When the following operations are executed, the following Confirm Message is popped up before the memory card program file is modified.



1. Delete folder or program file
2. Rename folder or program file
3. Drop-in program file
4. Add folder

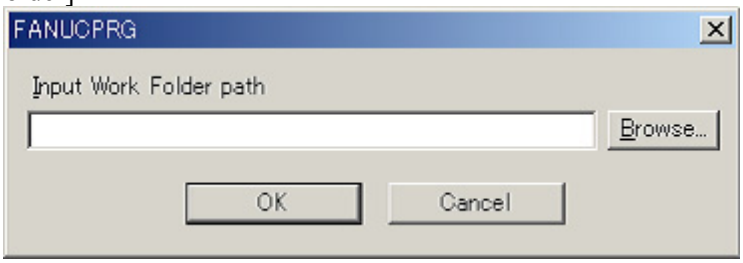
If the [OK] button is pushed, the operation advances.
If the [Cancel] button is pushed, the operation is abandoned.

When the [Hide Confirm Message] in menu bar is checked, the Confirm Message is not displayed and the operation advances at once. The default setting is the Confirm Message displayed.

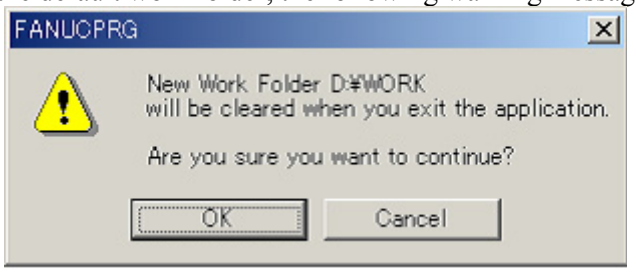
[Ignore Error Code]

When the [Ignore Error Code] in menu bar is checked, program file which includes unusable characters can be dropped in. The unusable characters in the program file will be ignored and will not be written into the memory card program file.

[Change Work Folder]

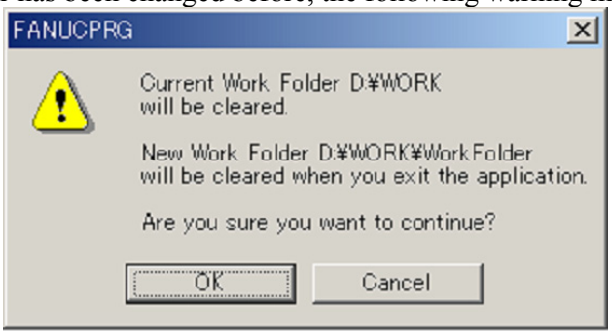


Work folder is used for temporarily keeping the dropped out files. If work folder has no enough free space, Drop-out will not be executed. To avoid this, you can check this option and change the work folder to an area where there is enough free space. With default setting, work folder [temp] will be created on the same place of this PC tool "FANUCPRG.exe".
If you changed the default work folder, the following warning message is popped up.



If the [OK] button is pushed, the operation advances.
If the [Cancel] button is pushed, the operation is abandoned.

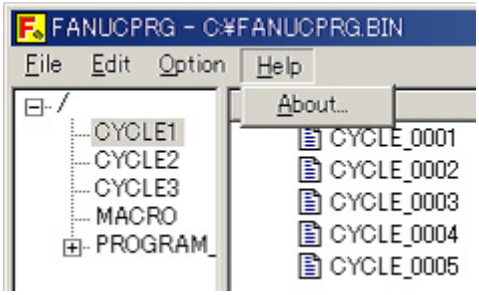
If the work folder has been changed before, the following warning message is popped up.



If the [OK] button is pushed, the operation advances.
If the [Cancel] button is pushed, the operation is abandoned.

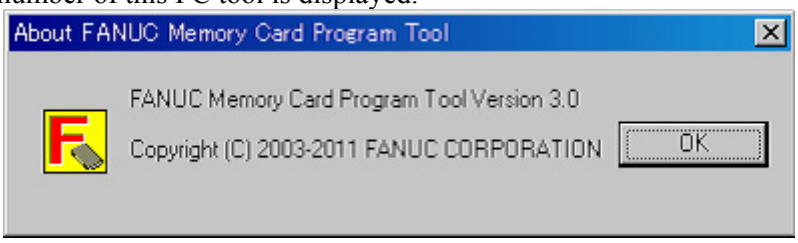
⚠ WARNING
1 At this PC tool terminated, all files in the work folder are deleted.
2 During this PC tool executing, do not access the files in the work folder. If the access is done, there is no assurance of normal operation.

Help menu



[About...]

Version number of this PC tool is displayed.



- Mouse Operation

[Drop-in and Drop-out]

- Drop-in from Explorer
  NC program can be added by dropping files including the NC files into the List view window of this PC tool from Explorer

  NC program name and update time is the same as the Dropped-in files.

  If "Oxxxxxxxx" or "<xxxx>" exists at the top of files, These "Oxxxxxxxx" and "<xxxx>" become NC program name. If not exist, each file name becomes NC program name.

Examples			
Name of external file	**Top of external file**	**Name of internal file**	**Program number**
O1234	N10G00	O1234	1234
O123N10G00	N10G00	O123N10G00	Not an O number program
test.txt	O1234N10G00	O1234	1234
test.txt	<O1234>	O1234	1234
test.txt	<O1234N10>	O1234N10	Not an O number program
O1234	<O1234N10>	O1234N10	Not an O number program
O001234	N10G00	O1234	1234
O001234N10G00	N10G00	O001234N10G00	Not an O number program
test.txt	O001234	O1234	1234
test.txt	<O001234>	O1234	1234
test.txt	<O001234N10G00>	O001234N10G00	Not an O number program

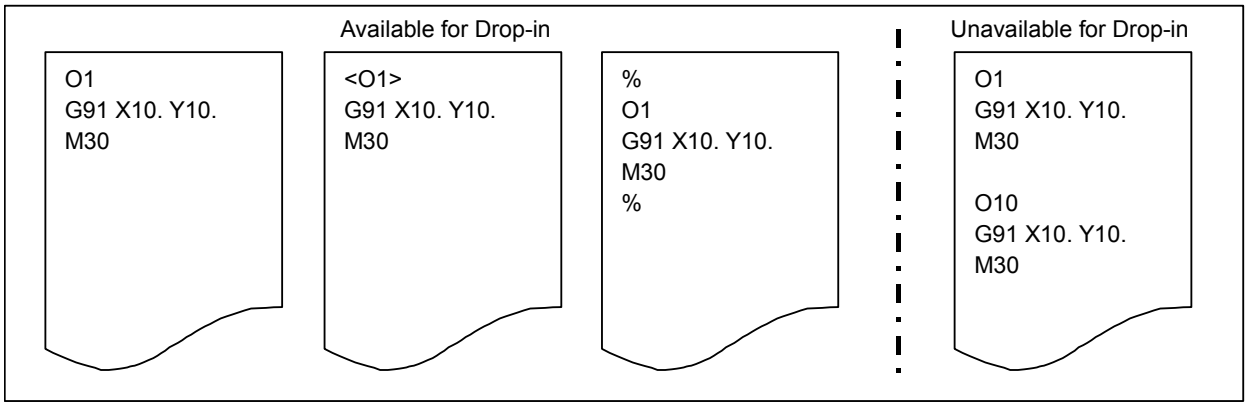
NOTE

1 For naming program file, please refer to the following chapter "Naming rules of Program file".
2 For usable characters in Program file, please refer to the following chapter "Rules of characters in Program file".
3 The updating time of program file is available from 1997 to 2037.

This PC tool can check contents of dropped-in program file according to "Rules of characters in Program file". However, this PC tool does not check grammar of NC program.

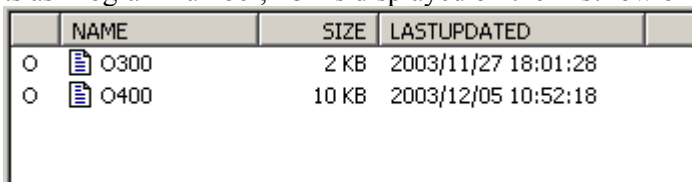
The program file can only have one NC program. Therefore, the NC program output from CNC cannot be dropped in directly.

Example of Program



NOTE
1 If the same named program file exists, another same named program file cannot be dropped in.
2 If free space on the memory card program file is lacking, new program file cannot be dropped in.
3 If the NC program name is not compliant with "Naming rules of Program file", the program file can not be dropped in.

If the File name acts as Program number, "O" is displayed on the first row of list view.



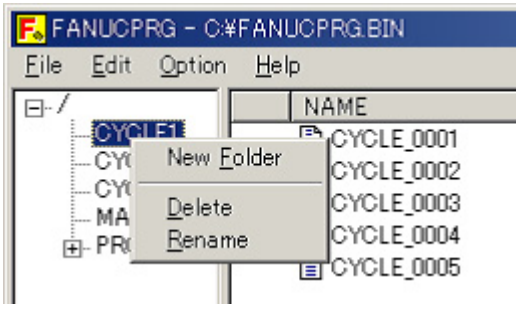
- Drop-out from List view window
  Drop-out from list view of this PC tool to Explorer is available.

NOTE
Do not drop out to Work folder. If dropped out to Work folder, this PC tool cannot continue to function normally.

- Pop-up menu

Pop-up menu is displayed by clicking the right mouse button.

- Focus on Tree view

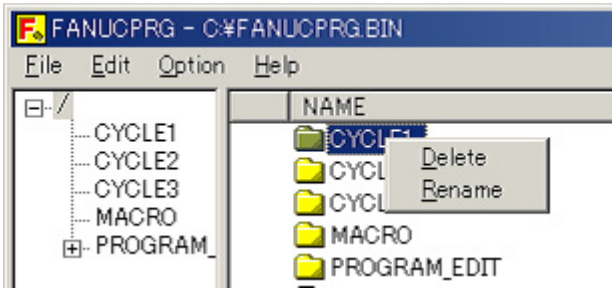


Clicking "New Folder", a new folder is created on selected folder.
Clicking "Delete", the selected folder is deleted.

Clicking "Rename", the selected folder is renamed.

If clicking on root folder, "Delete" and "Rename" are not activated.

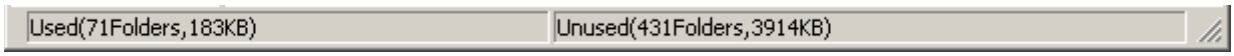
- Focus on List view



- Clicking "Delete", the selected folder or program file is deleted.
- Clicking "Rename", the selected folder or program file is renamed.

- Display of free space on the memory card program file ("FANUCPRG.BIN")

Number of used folder, size of used space, number of unused folder, and size of free space are displayed on status bar in the lower portion of the screen.

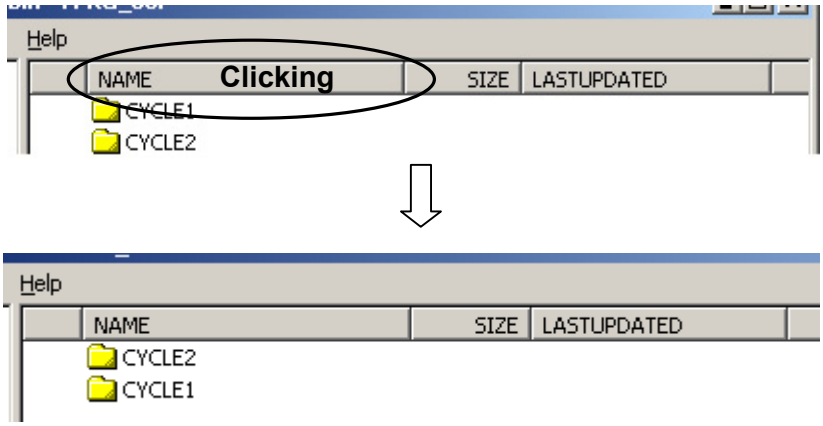


When a new memory card program file is created, two reserved folder are also created. Therefore, the number of used folder is two. However it does not show that number of unused folder is reduced.

The display of status bar is renewed at creating or deleting a folder, dropping-in from Explorer, and deleting a program file.

- Sorting list view of the memory card program file

When a column is being clicked, the list view of the memory card program file is being sorted by the column key in ascending or descending order.
The initial display is sorted by NAME key in ascending order.



I.2 NAMING RULES

Overview

Naming rules of folder and program file are described as follows.

I.2.1 Naming Rules of Program File

Here are Naming rules of Program file:

- Program file name can have a maximum of 32 characters.
- Program file name can have following characters.
  Alphabet(Upper and lower case letter), numeric character,
  "-"(minus), "+"(plus), "_"(under bar), "."(dot)
  "." and ".." can not be used since these are reserved for system use.

- The File name acts as Program number

When the File name is "O"+ 1-99999999, the file name acts as Program number.

Example)

"O123"	Program number 123
"O1"	Program number 1
"O3000"	Program number 3000
"O99999999"	Program number 99999999
"O0123"	Program number 123

The file name does not act as Program number, and can be dropped in:

"ABC"	Top of character is not upper case letter of "O".
"o123"	Top of character is not upper case letter of "O".
"O0123XY"	Characters following the "O" contains alphabet.

The file name does not act as Program number, and cannot be dropped in:

"O123456789"	Numeric characters exceed 8 digits.

NOTE
1 Program file name cannot be repeated in a Folder.
2 If program file name starts with "O" and the next eight characters are all numeric characters, the "0" (zero) after the "O" will be deleted.

I.2.2 Naming Rules of Folder

Here are Naming rules of Folder:

- Folder name can have a maximum of 32 characters.
- Folder name can have following characters.
  Alphabet(Upper and lower case letter), numeric character,
  "-"(minus), "+"(plus), "_"(under bar), "." (dot)
  "." and ".." can not be used since these are reserved for system use.

NOTE
Folder name cannot be repeated in a Folder.

I.3 RULES OF CHARACTERS IN PROGRAM FILE

Overview

Words in parentheses "( )" in Program file are treated as comments.
The mark of comment start "(" is named "Control-out".
The mark of comment end ")" is named "Control-in".
"Control-out" and "Control-in" must make a pair. The order is 1st - "Control-out" and 2nd -"Control-in".
And nested parentheses is not available.

NOTE
1 When a program file is dropped in, space code (0x20 SPC) , tab code (0x09 HT) , carriage return code(0x0d CR) and percent code(0x25 %) is deleted. If "%" is found in Control-in, characters between "%" and next "LF" (0x0a) is deleted.
2 The front of program number ":" is changed to "O"(O as in Oscar) while the program file dropped in.
3 The program file can only have one NC program.

I.3.1 Usable Characters in Program File

- Usable characters in Control-in

List of ANSI(ASCII) codes of usable characters(hexadecimal form)

Code	Character	Code	Character	Code	Character	Code	Character
0a	LF	3f	?	58	X	74	t
23	#	40	@	59	Y	75	u
26	&	41	A	5a	Z	76	v
28	(	42	B	5b	[	77	w
29	)	43	C	5d	]	78	x
2a	*	44	D	5f	_	79	y
2b	+	45	E	61	a	7a	z
2c	,	46	F	62	b		
2d	-	47	G	63	c		
2e	.	48	H	64	d		
2f	/	49	I	65	e		
30	0	4a	J	66	f		
31	1	4b	K	67	g		
32	2	4c	L	68	h		
33	3	4d	M	69	i		
34	4	4e	N	6a	j		
35	5	4f	O	6b	k		
36	6	50	P	6c	l		
37	7	51	Q	6d	m		
38	8	52	R	6e	n		
39	9	53	S	6f	o		
3a	:	54	T	70	p		
3c	<	55	U	71	q		
3d	=	56	V	72	r		
3e	>	57	W	73	s		

NOTE
In the Control-in, "O", ":", and "<" can not be used at top of the line except for the 1st line.

- Usable characters in Control-out(characters in parentheses)

List of ANSI(ASCII) codes of usable characters(hexadecimal form)

Code	Character	Code	Character	Code	Character	Code	Character
0a	LF	3c	<	55	U	71	q
20	SPC	3d	=	56	V	72	r
22	"	3e	>	57	W	73	s
23	#	3f	?	58	X	74	t
24	$	40	@	59	Y	75	u
26	&	41	A	5a	Z	76	v
27	'	42	B	5b	[	77	w
2a	*	43	C	5d	]	78	x
2b	+	44	D	5f	_	79	y
2c	,	45	E	61	a	7a	z
2d	-	46	F	62	b		
2e	.	47	G	63	c		
2f	/	48	H	64	d		
30	0	49	I	65	e		
31	1	4a	J	66	f		
32	2	4b	K	67	g		
33	3	4c	L	68	h		
34	4	4d	M	69	i		
35	5	4e	N	6a	j		
36	6	4f	O	6b	k		
37	7	50	P	6c	l		
38	8	51	Q	6d	m		
39	9	52	R	6e	n		
3a	:	53	S	6f	o		
3b	;	54	T	70	p		

I.4 ERROR MESSAGE AND NOTE

Error may occur when using this application, hereafter explains the error messages and gives relative instructions.

I.4.1 List of Error Message

When an error occurred, the error message box is displayed as follows.

Message	Remarks
Failed to open the file you specified.	If failed to open once again, the file may be broken.
Failed to read or write to the specified file.	
There is insufficient disk space.	There is no enough free space to create a new memory card program file or to put drop out files to work folder. In the latter case, please refer to chapter "Menu" [Change Work Folder]
File name is not correct.	Please refer to the chapter "Naming rules of Program file".
Input name is already exists.	Please input another name.
File name is already exists.	Please input another name.
Input name is not correct.	Please refer to the chapter "Naming rules".

Message	Remarks
Please input an integer between 2 and 2048	The size of the memory card program file is available from 2Mbyte to 2048Mbyte.
An illegal character is included in the specified file.	Please refer to the chapter "Rules of characters in Program file".
Last update time of the specified file is unsupported.	The updating time of program file is available from 1997 to 2037.
The memory card program file you specified cannot be identified.	The specified file format is not the memory card program file
There is insufficient free folder.	Please delete unnecessary program files or folders.
There is insufficient free program space.	Please delete unnecessary program files.
Folder deeper than 7 cannot be created.	
Root folder cannot be deleted.	
Root folder cannot be renamed.	
Failed to create work folder.	Please check whether the [temp] work folder is able to create or not. In default, the [temp] work folder is created on the same place of this PC tool "FANUCPRG.exe".
Process has been cancelled.	
The specified work folder is not found.	The Drop-out cannot be executed. Terminate this PC tool. And check the setting of Work Folder on the Option dialogue window.
Only one instance of this application can be executed.	Double activation of this PC tool cannot be executed.

I.4.2 Note

- Folder and Program Numbers

Folder and Program Numbers can be selected among 63 / 500 / 1000.
For selecting the Numbers of 500 or 1000, the option "Registered programs expan. On the memory card" is needed in CNC side.

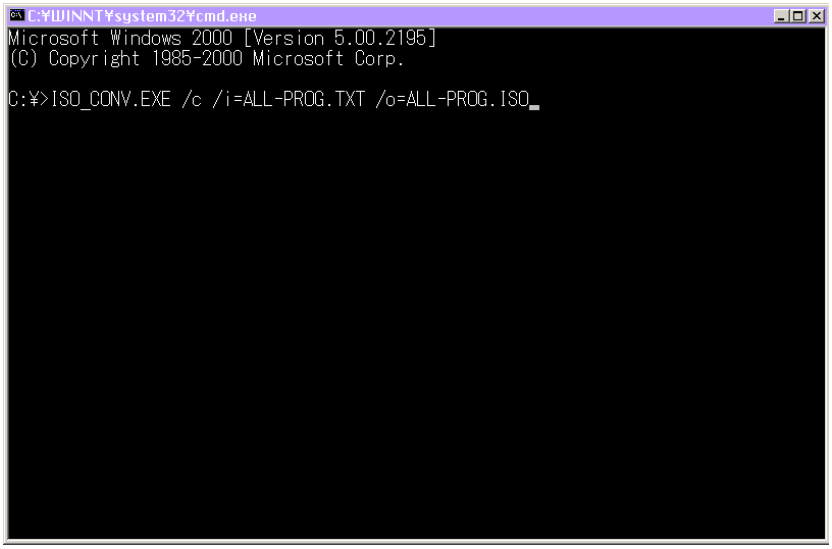
J ISO/ASCII CODE CONVERSION TOOL

Overview

FANUC ISO Converter is a tool that converts a file created or externally output with ASCII code to ISO code format.
This tool runs on Windows 2000, Windows XP, and Windows Vista.
This tool can be used in two modes, CUI and GUI.

CUI

Start the executable file from the command prompt.



Specify the following arguments when starting the tool.

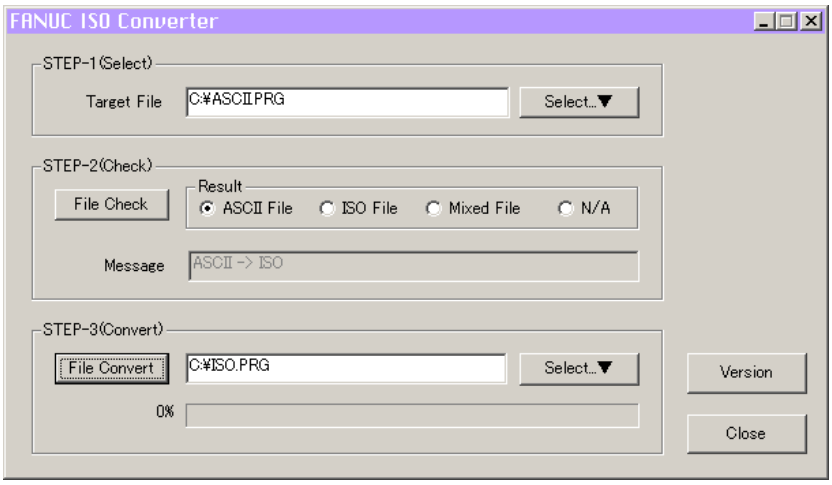
/c : Specify that the tool runs in command line mode.
When /c is omitted, the tool runs in GUI mode.
/i= : Specify the name of an input file to be converted.
/o= :Specify the name of the converted file.

Example
C:¥>ISO_CONV.EXE /c /i=ALL-PROG.TXT /o=ALL-PROG.ISO

Multiple files can be converted at a time by listing them in a batch file.

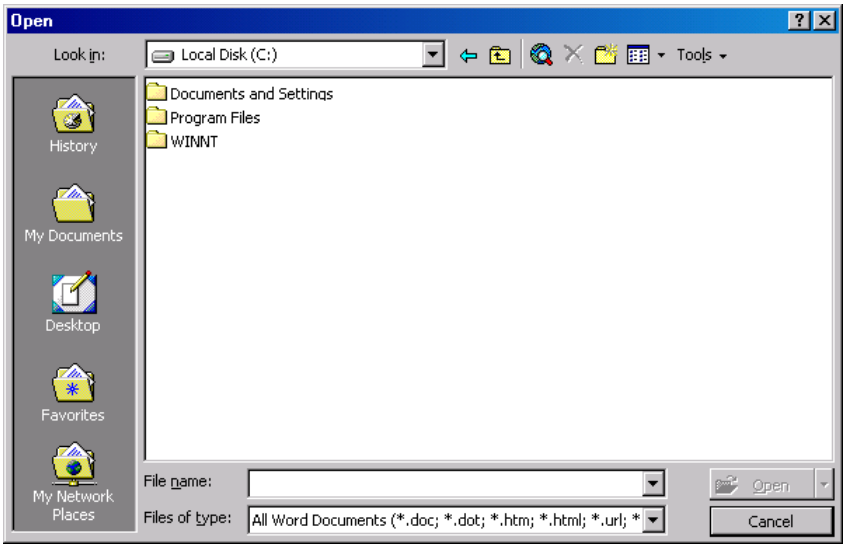
GUI

When you double-click the icon, the following screen appears, allowing you to select and convert a file.



Conversion procedure

1. Step 1
   In [Target File], specify a file you want to convert.
   When you click the [Select...▼] button, a file selection dialogue appears, allowing you to select a file.

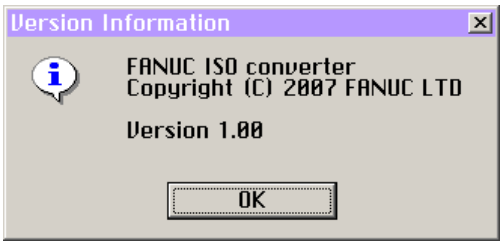


2. Step 2
   After you specify the file name in step 1, you can determine the code of the specified file by clicking the [File Check] button. When you select the file by clicking the [Select...▼] button, the code is automatically determined.
   The file is determined as an ASCII file, ISO file, or ASCII/ISO-mixed file, or is indeterminable. In [Message], an additional message appears depending on the determination result.
   If the file is determined as an ASCII/ISO-mixed file, it cannot be converted, so you cannot proceed to the next step. The [File Convert] button is disabled and you cannot click it. If file contains only characters represented by the code values common to ASCII and ISO, the code of the file is indeterminable. This file need not be converted, so the [File Convert] button is disabled and you cannot click it.

3. Step 3
   When you specify the name of a converted file and click the [File Convert] button, the converted file is created. When the original file is an ASCII file, an ISO file is created; when the original is an ISO file, an ASCII file is created.
   You can also click the [Select...▼] button and specify the name of the converted file from a file selection dialogue.

Other

When you click the [Version] button, the software version information is displayed as shown below.



INDEX

<Number>

10.4" LCD CNC Display Panel (12.1"/15"/19" LCD CNC Display Panel) 940
3-DIMENSIONAL CIRCULAR INTERPOLATION 122
3-DIMENSIONAL COORDINATE SYSTEM CONVERSION 284
3-DIMENSIONAL CUTTER COMPENSATION 850
3-DIMENSIONAL MANUAL FEED 984
5-AXIS MACHINING FUNCTION 682
8.4" LCD CNC Display Panel 939

<A>

ABSOLUTE AND INCREMENTAL PROGRAMMING 177
Absolute multiple command 815
ACTION TO BE TAKEN WHEN A PROBLEM OCCURRED 1814
ACTIVE BLOCK CANCEL FUNCTION 1101
Actual Feedrate Display 1351
Actual Feedrate Display (15/19-inch Display Unit) 1363
Addition of Workpiece Coordinate System Pair (G54.1 or G54) 165
AI CONTOUR CONTROL FUNCTION I AND AI CONTOUR CONTROL FUNCTION II (G05.1) 561
ALARM AND SELF-DIAGNOSIS FUNCTIONS 1161
Alarm Display 935,1161
ALARM HISTORY DISPLAY 1163
ALARM LIST 2184
Altering a Word 1268
Angular Axis Control 1001
Animation 1767
ANIMATION GRAPHIC screen 1770
ARITHMETIC AND LOGIC OPERATION 440
ARITHMETIC AND LOGICAL OPERATION 514
Automatic Coordinate System Setting 167
Automatic Corner Override 137
AUTOMATIC INSERTION OF SEQUENCE NUMBERS 1259
AUTOMATIC OPERATION 927,1007
Automatic override for inner corners (G62) 137
AUXILIARY FUNCTION 248
AUXILIARY FUNCTION (M FUNCTION) 248
AUXILIARY FUNCTION OUTPUT BLOCK REVERSE MOVEMENT FOR MANUAL HANDLE RETRACE 1126
Auxiliary Function Output in Program Restart Function 1047
Available Keys 1281
Axis Control by PMC 1001
AXIS CONTROL COMMAND 523
AXIS CONTROL FUNCTIONS 607
Axis status display 1148
Axis Synchronization Control 999
AXIS SYNCHRONOUS CONTROL 607

<B>

Background Editing 1386
Background Editing (15/19-inch Display Unit) 1426
BACKING UP VARIOUS DATA ITEMS 1815
Battery in the PANEL *i* (3 VDC) 1821
BRANCH AND REPETITION 450
Built-in 3D Interference Check 1576

<C>

Canceling Spindle Positioning 194
CHANGING CURRENT FOLDER 1309
CHANGING FILE ATTRIBUTES 1314
CHANGING FOLDER ATTRIBUTES 1311
Changing Workpiece Coordinate System 161
CHARACTER-TO-CODES CORRESPONDENCE TABLE 2177
Check by Running the Machine 928
CHECKING BY DIAGNOSTIC DISPLAY 1164
CHOPPING FUNCTION 664
CHOPPING FUNCTION BY FLEXIBLE SYNCHRONOUS CONTROL 673
CIRCULAR INTERPOLATION (G02, G03) 46
CIRCULAR THREADING (G35, G36) 111
CODES AND RESERVED WORDS USED IN CUSTOM MACROS 486
Color Setting Screen 1635
Color Setting Screen (15/19-inch Display Unit) 1679
COMMAND FOR MACHINE OPERATIONS - AUXILIARY FUNCTION 23
Command type selection screen 1405,1445
COMMON MEMORY BETWEEN EACH PATH 917
COMPENSATION FUNCTION 295
Condition Transition (ZEDGE Statement) 517
Conditional Branch (IF Statement) 451
Conditional Branch (ZONCE Statement) 516
Confirmation of a data update during the data setting process 1146
Confirmation of incremental input 1144
Confirmation of the deletion of all data 1146
Confirmation of the deletion of the program 1145
Confirmation of the start from a middle block 1149
CONSTANT SURFACE SPEED CONTROL (G96, G97) 187
CONTINUOUS HIGH-SPEED SKIP FUNCTION 118
CONTROL ON REAL TIME MACRO COMMANDS 515
CONTROLLED AXES 27
Controlled axis configuration example 651
COORDINATE SYSTEM 154
Coordinate System on Part Drawing and Coordinate System Specified by CNC - Coordinate System 13
COORDINATE VALUE AND DIMENSION 177
Copy 1284
Copy and movement between different devices 1318

Copying and Moving Files between Devices.....1287
CREATING A FOLDER.....1308
CREATING PROGRAMS.....1258
CREATING PROGRAMS IN TEACH IN MODE (PLAYBACK).....1260
CREATING PROGRAMS USING THE MDI PANEL.....1258
Creation.....1285
Current Position Display.....934
CURSOR MOVEMENT LIMITATIONS ON PROGRAM EDITING.....1276
CUSTOM MACRO.....380
Custom Macro Variables.....512
Customization of Tool Management Data Display.....217
Cut.....1284
Cutter Compensation in Composite Type Machine.....881
Cutter Compensation in Table Rotation Type Machine876
Cutter Compensation in Tool Rotation Type Machine 852
CUTTING FEED.....129
CUTTING FEEDRATE CONTROL.....136
CUTTING POINT COMMAND.....776
CUTTING POINT INTERPOLATION FOR CYLINDRICAL INTERPOLATION (G07.1).....71
CUTTING SPEED - SPINDLE FUNCTION.....21
Cylinder setting screen.....1602
Cylindrical Interpolation.....66
CYLINDRICAL INTERPOLATION (G07.1).....66
Cylindrical Interpolation by Plane Distance Command.69

<D>

DATA INPUT/OUTPUT.....1169
Data range check.....1150
DATA TYPE.....2146
DECIMAL POINT PROGRAMMING.....182
Default Folders.....257
Definition of the Custom Macro Screen.....553
Definition of the Pattern Menu Screen.....551
DEFINITION OF THE SCREEN.....550
DEFINITION OF WARNING, CAUTION, AND NOTE.....s-1
Deleting a Block.....1271
DELETING A FILE.....1313
DELETING A FOLDER.....1311
Deleting a Word.....1268
Deleting All Programs.....1275
DELETING BLOCKS.....1271
Deleting Multiple Blocks.....1271
Deleting One Program.....1275
DELETING PROGRAMS.....1275
Description of commands compatible with those for a hobbing machine (G80, G81).....648
DESCRIPTION OF PARAMETERS.....1827
Details.....1296,1790,1801
Details of Functions.....494
Details of the tilted working plane data setting screen.....1409,1449
DIAMETER AND RADIUS PROGRAMMING.....183
DIAMETER AND RADIUS SETTING SWITCHING FUNCTION.....184
Direct Input of Workpiece Origin Offset Value Measured.....1465
Direct Input of Workpiece Origin Offset Value Measured (15/19-inch Display Unit).....1524
Disabling Life Count.....246
DISPLAY.....933
Display of 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount).....1357
Display of 3-dimensional Manual Feed (Tool Tip Coordinates, Number of Pulses, Machine Axis Move Amount) (15/19-inch Display Unit).....1369
Display of Run Time and Parts Count.....1353
Display of Run Time and Parts Count (15/19-inch Display Unit).....1365
Display of updated modal information.....1147
Display setting screen.....1614
Displaying and Entering Setting Data.....1458
Displaying and Entering Setting Data (15/19-inch Display Unit).....1517
Displaying and Setting Custom Macro Common Variables.....1466
Displaying and Setting Custom Macro Common Variables (15/19-inch Display Unit).....1525
DISPLAYING AND SETTING DATA.....930
Displaying and setting magazine screen.....1472
Displaying and setting magazine screen (15-inch display unit).....1531
Displaying and Setting Parameters.....1627
Displaying and Setting Parameters (15/19-inch Display Unit).....1672
Displaying and Setting Pattern Data Inputs.....1512
Displaying and Setting Pattern Data Inputs (15/19-inch Display Unit).....1573
Displaying and Setting Real Time Custom Macro Data.....1468
Displaying and Setting Real Time Custom Macro Data (15/19-inch Display Unit).....1527
Displaying and Setting Run Time, Parts Count, and Time.....1462
Displaying and Setting Run Time, Parts Count, and Time (15/19-inch Display Unit).....1521
Displaying and setting the FSSB axis setting screen..1652
Displaying and setting the FSSB axis setting screen (15/19-inch display unit).....1696
Displaying and setting the FSSB servo amplifier setting screen.....1651
Displaying and setting the FSSB servo amplifier setting screen (15/19-inch display unit).....1695
Displaying and setting the FSSB spindle amplifier setting screen.....1651
Displaying and setting the FSSB spindle amplifier setting screen (15/19-inch display unit).....1695
Displaying and setting the machining parameter tuning screen.....1655

Displaying and setting the machining parameter tuning screen (15/19-inch display unit) 1699
Displaying and setting the servo setting screen....1652
Displaying and setting the servo setting screen (15/19-inch display unit) 1696
Displaying and setting the servo tuning screen 1654
Displaying and setting the servo tuning screen (15/19-inch display unit) 1698
Displaying and Setting the Software Operator's Panel1469
Displaying and Setting the Software Operator's Panel (15/19-inch Display Unit)....1528
Displaying and setting the spindle tuning screen 1654
Displaying and setting the spindle tuning screen (15/19-inch display unit) 1698
Displaying and Setting the Workpiece Origin Offset Value 1464
Displaying and Setting the Workpiece Origin Offset Value (15/19-inch Display Unit) 1523
Displaying and Setting Tool Life Management Data.1500
Displaying and Setting Tool Life Management Data (15/19-inch Display Unit)....1561
Displaying and setting tool management screen 1473
Displaying and setting tool management screen (15-inch display unit)....1532
Displaying and Setting Workpiece Setting Error Compensation Data 1511
Displaying and Setting Workpiece Setting Error Compensation Data (15/19-inch Display Unit) 1572
Displaying and Switching the Display Language 1488
Displaying and Switching the Display Language (15/19-inch Display Unit)....1549
Displaying Memory Data....1644
Displaying Memory Data (15/19-inch Display Unit).1687
Displaying Program Being Executed in Backward Movement....1124
Displaying the executed block 1373,1417
Displaying the menu screen and selecting a menu item....1646
Displaying the menu screen and selecting a menu item (15/19-inch display unit) 1690
Displaying the Program Name 1739
Displaying the Program Number, Program Name, and Sequence Number....1734
Displaying the Program Number, Program Name, and Sequence Number (15/19-inch Display Unit) 1738
DISPLAYING THE PROGRAM NUMBER/NAME, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION 1734
Displaying the Status and Warning for Data Setting or Input/Output Operation 1735
Displaying the Status and Warning for Data Setting or Input/Output Operation (15/19-inch Display Unit) 1740
Displaying the total life of tools of the same type....1481
Displaying the total life of tools of the same type (15-inch display unit) 1540
DISTANCE CODED LINEAR SCALE INTERFACE997
Distance Coded Rotary Encoder 999
DNC OPERATION....1012
Drawing coordinate system setting screen 1615
DRY RUN....1109
DWELL 144
DWELL/AUXILIARY FUNCTION TIME OVERRIDE....1107
DYNAMIC GRAPHIC DISPLAY....1754

<E>

Each tool Data Screen....226,1478
Each tool data screen (15-inch display unit) 1538
EDIT DISABLE ATTRIBUTE AND DISPLAY DISABLE ATTRIBUTE....1263
Edit of multi-path program....1804
EDITING A PROGRAM....930,1376
Editing a Program (15/19-inch Display Unit) 1417
EDITING OF CUSTOM MACROS 1275
EDITING PROGRAM CHARACTERS....1279
EDITING PROGRAMS....1263
ELECTRONIC GEAR BOX....626
Electronic Gear Box 2 Pair 645
Electronic Gear Box Automatic Phase Synchronization....639
EMBEDDED ETHERNET OPERATIONS 1249
EMERGENCY STOP 1134
ERROR MESSAGE AND NOTE....2277
Exact Stop (G09, G61), Cutting Mode (G64), Tapping Mode (G63)....137
Examples....895
EXPANSION OF AXIS MOVE COMMAND IN TOOL CENTER POINT CONTROL 764
EXPANSION OF THE WAY TO SET 5-AXIS MACHINING FUNCTION PARAMETERS....900
EXPLANATION 542
EXPLANATION OF OPERATION 546
Explanation of Operations....2267
EXPLANATION OF THE MDI UNIT....944
EXPONENTIAL INTERPOLATION (G02.3, G03.3) ..81
EXTENSION OF THE MANUAL HANDLE RETRACE FUNCTION....1130
EXTERNAL I/O DEVICES....957
External Operator Message History 1715
EXTERNAL OUTPUT COMMANDS....486
EXTERNAL SUBPROGRAM CALL (M198)....1018
EXTERNAL SUBPROGRAM CALLS USING THE DATA SERVER AVAILABLE IN MULTI-PATH SYSTEMS....1022

<F>

FANUC DOUBLE-BYTE CHARACTER CODE TABLE....2178
FEED FUNCTIONS 127
FEED-FEED FUNCTION 11
FEEDRATE INSTRUCTION ON IMAGINARY CIRCLE FOR A ROTARY AXIS....140
FEEDRATE OVERRIDE 1106
FIGURE COPYING (G72.1, G72.2) 279
Figure setting menu screen....1581

File Format and Error Messages 1248
FIVE AXES DISPLAY IN ONE SCREEN FOR THE 8.4-INCH DISPLAY UNIT.....1722
FLOATING REFERENCE POSITION RETURN (G30.1) 152
Folder Attributes 257
Folder Configuration.....255
Folder for Subprogram/Macro Calls 1324
FOLDER MANAGEMENT.....1319
Folder tree display.....1383
Folder tree display (15/19-inch display unit) 1424
FOLDERS.....255
FTP File Transfer Function 1249
Function Keys 948
FUNCTION KEYS AND SOFT KEYS.....947
Functions that are Used When Data is Set 1142
Functions that are Used when the Program is Executed.....1147
FUNCTIONS TO SIMPLIFY PROGRAMMING 279

<G>

G CODE LIST IN THE LATHE SYSTEM 36
G CODE LIST IN THE MACHINING CENTER SYSTEM 32
G53, G28, G30, and G30.1 Commands in Tool Length Compensation Mode.....300
GENERAL 3,9,925
General Screen Operations.....947
General specifications of the tilted working plane indexing.....790
GENERAL WARNINGS AND CAUTIONS s-2
GOTO Statement Using Stored Sequence Numbers 450
GRAPHIC DISPLAY 1743
GRAPHIC FUNCTION 1743
GRAPHIC PARAMETER (DYNAMIC GRAPHIC) screen.....1754,1767

<H>

Heading a Program.....1267
HELICAL INTERPOLATION (G02, G03).....50
HELICAL INTERPOLATION B (G02, G03) 51
HIGH PRECISION OSCILLATION FUNCTION 674
HIGH SPEED PROGRAM CHECK FUNCTION.....1111
HIGH-SPEED BINARY PROGRAM OPERATION ..578
HIGH-SPEED CUTTING FUNCTIONS.....561
HIGH-SPEED CYCLE MACHINING 577
HIGH-SPEED SKIP SIGNAL (G31).....117
HIGH-SPEED SMOOTH TCP 739
How to Indicate Command Dimensions for Moving the Tool (Absolute and Incremental Programming).....18
How to View the Position Display Change without Running the Machine 930
HYPOTHETICAL AXIS INTERPOLATION (G07) ..108

<I>

Improvement of Tool compensation for Tool Retract and Recover.....622
INCH/METRIC CONVERSION (G20, G21) 179
INCLINED ROTARY AXIS CONTROL.....847
INCORRECT THREADED LENGTH.....2167
INCREMENT SYSTEM.....28
INCREMENTAL FEED 962
Incremental multiple command.....817
INDIRECT AXIS ADDRESS SPECIFICATION.....447
Information Display in High-speed Smooth TCP 758
Input and output setting data for 3D interference check 1617
Input data range check 1143
Input Mode.....1282
Input of Customize Data with the Decimal Point.....223
Input/output format of 3-dimensional error compensation data 1190
Input/output format of operation history signal selection data.....1203
Input/output format of pitch error compensation data 1187
Input/output format of three-dimensional rotary error compensation data 1194
Input/output format of workpiece setting error values1221
Input/output of Multi-path program 1805
INPUT/OUTPUT ON EACH SCREEN 1173
INPUT/OUTPUT ON THE ALL IO SCREEN.....1224
Inputting 3-dimensional error compensation data.....1188
Inputting a program.....1174
Inputting and Outputting 3-dimensional Error Compensation Data 1188
Inputting and Outputting a Program.....1174
Inputting and Outputting All Tool Management Data at a Time 1244
Inputting and Outputting Custom Macro Common Variables 1196
Inputting and Outputting Machine Configuration Data 909
Inputting and outputting of O8-digit 1177
Inputting and Outputting Offset Data.....1180,1232
Inputting and Outputting Operation History Data 1200,1238
Inputting and Outputting Parameters 1178,1231
Inputting and Outputting Pitch Error Compensation Data 1185
Inputting and Outputting Three-dimensional Rotary Error Compensation Data.....1192
Inputting and Outputting Tool Life Management Data 1222
Inputting and Outputting Tool Management Data 1204,1239
Inputting and Outputting Workpiece Coordinates System Data 1199,1237
Inputting and Outputting Workpiece Setting Error Compensation Value 1219,1247
Inputting custom macro common variables 1196
Inputting customize data displayed as tool management data.....1211
Inputting decimal point position data of customize data.....1215
Inputting magazine data.....1206
Inputting name data of customize data.....1210

Inputting offset data 1180
Inputting operation history signal selection data....1201
Inputting parameters....1178
Inputting pitch error compensation data....1185
Inputting spindle waiting position name data....1213
Inputting three-dimensional rotary error compensation data 1192
Inputting tool geometry data 1217
Inputting tool life management data....1222
Inputting tool life status name data 1208
Inputting tool management data 1204
Inputting values on the workpiece setting error compensation screen....1219
Inputting workpiece coordinate system data 1199
Inputting/Outputting a Program 1227
Inputting/Outputting all Programs and Folders 1229
Inputting/Outputting Custom Macro Common Variables 1235
Inputting/Outputting Pitch Error Compensation Data 1234
Inserting a Word....1267
INSERTING, ALTERING AND DELETING A WORD....1264
Interference Check and Interference Avoidance 886
Interference check valid figure selection screen 1607
Internal circular cutting feedrate change 139
INTERPOLATION FUNCTIONS 40
INTERRUPTION TYPE CUSTOM MACRO 492
ISO/ASCII CODE CONVERSION TOOL 2279

<J>

JOG FEED (JOG) 960

<K>

KEYS AND PROGRAM ENCRYPTION 1290

<L>

Leading edge offset....869
LIMITATION 535,1416,1456
Limitations 1789,1800,1810
Line Number Display....1282
Line Search 1285
LINEAR INTERPOLATION (G01) 44
LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL)....1002
List of Error Message....2277
LIST OF FUNCTIONS AND PROGRAM FORMAT 2152
List of Functions of PC Tool....2266
LOAD METER SCREEN....1729
LOCAL COORDINATE SYSTEM....169
Local variables....514

<M>

M Code Group Check Function 252
M CODE GROUPING FUNCTION....250
MACHINE CONFIGURATION SELECTING FUNCTION....903
Machine Configuration Selecting Screen....903
MACHINE COORDINATE SYSTEM....154
MACHINE LOCK AND AUXILIARY FUNCTION LOCK....1105
MACHINING CONDITION SELECTING FUNCTION....575
Machining Level Selection 1498
Machining Level Selection (15/19-inch Display Unit) 1558
Machining Parameter Tuning....1636
Machining Parameter Tuning (15/19-inch Display Unit) 1680
Machining parameter tuning (AI contour) 1636,1680
Machining parameter tuning (nano smoothing) 1642,1685
MACHINING QUALITY LEVEL ADJUSTMENT ...575
Machining Quality Level Selection 1499
Machining Quality Level Selection (15/19-inch Display Unit)....1559
MACRO CALL....456,521
MACRO CALL ARGUMENT FOR AXIS NAME EXPANSION 482
Macro Call Using a G Code 469
Macro Call Using a G Code (Specification of Multiple Definitions) 470
Macro Call Using a G Code with a Decimal Point (Specification of Multiple Definitions) 471
Macro Call Using an M Code 471
Macro Call Using an M Code (Specification of Multiple Definitions)....473
MACRO STATEMENTS AND NC STATEMENTS..449
Making of machining programs 1802
MANUAL ABSOLUTE ON AND OFF 966
MANUAL HANDLE FEED....963
MANUAL HANDLE INTERRUPTION 1023
MANUAL HANDLE RETRACE....1113
MANUAL HANDLE RETRACE FUNCTION FOR MULTI-PATH 1126
Manual Interruption of 3-dimensional Coordinate System Conversion....1029
MANUAL INTERVENTION AND RETURN....1089
MANUAL LINEAR/CIRCULAR INTERPOLATION 970
MANUAL NUMERICAL COMMAND....976
MANUAL OPERATION....925,959
MANUAL REFERENCE POSITION RETURN....959
Maximum incremental value check 1151
MAXIMUM STROKE....29
MDI OPERATION 1009
MEMORY OPERATION 1007
METHOD OF REPLACING BATTERY 1817
MIRROR IMAGE....1030
Modal Call: Call After the Move Command (G66) 464
Modal Call: Each Block Call (G66.1)....467
Modal Real Time Macro Command / One-shot Real Time Macro Command 503
Monitor menu screen 1576
Moving axis setting menu screen and moving axis setting screen....1609
MUITI-PATH CONTROL FUNCTION....911

MULTI-PATH EDITING FUNCTION 1296
Multi-path program folder.....1802
MULTI-PATH PROGRAM MANAGEMENT FUNCTION.....1801
Multiple command of tilted working plane indexing ...815
MULTIPLE M COMMANDS IN A SINGLE BLOCK 249
Multi-statement (ZDO...ZEND Statement).....519
MULTI-STEP SKIP (G31) 117

<N>

Name change of the multi-path program folder 1808
Name setting screen 1612
NAMES OF AXES 27
NAMING RULES.....2275
Naming Rules of Folder 2275
Naming Rules of Program File.....2275
NANO SMOOTHING 92
Next Block Display Screen 1385
Next Block Display Screen (15/19-inch Display Unit) 1424
NOMOGRAPHS.....2167
NORMAL DIRECTION CONTROL (G40.1,G41.1,G42.1).....311
Note.....1001,1778,2278
NOTES.....534
NOTES ON READING THIS MANUAL 6
NOTES ON VARIOUS KINDS OF DATA.....6
NUMBER OF CONTROLLED AXES.....27
NURBS INTERPOLATION (G06.2).....100
NURBS Interpolation Additional Functions 104

<O>

Object figure setting screen.....1582
Operating Monitor Display 1355
Operating Monitor Display (15/19-inch Display Unit) 1367
Operation.....1794
Operation confirmation function setting screen 1151
Operation level setting 1490
Operation level setting (15-inch display unit).....1550
Operation of inputting and editing for each program .1808
Operation Procedure for a Canned Cycle for Drilling 1087
OPERATIONAL DEVICES 937
OPTIMUM ACCELERATION/DECELERATION FOR RIGID TAPPING 579
OTHERS.....522
Outputting 3-dimensional error compensation data ...1190
Outputting a program 1176
Outputting custom macro common variables.....1197
Outputting customize data displayed as tool management data.....1212
Outputting decimal point position data of customize data 1216
Outputting magazine data.....1208
Outputting name data of customize data 1211
Outputting offset data.....1181
Outputting operation history data.....1201
Outputting operation history signal section data.....1202
Outputting parameters.....1179
Outputting pitch error compensation data 1186
Outputting spindle waiting position name data 1214
Outputting three-dimensional rotary error compensation data 1193
Outputting tool geometry data 1218
Outputting tool life management data 1223
Outputting tool life status name data.....1209
Outputting tool management data 1205
Outputting values on the workpiece setting error compensation screen.....1220
Outputting workpiece coordinate system data 1200
Overall Position Display 1349
Overall Position Display (15/19-inch Display Unit) ..1360
OVERTRAVEL.....1134
OVERVIEW 127,295,542,911,1296,1822
OVERWRITING FILES ON A MEMORY CARD/USB MEMORY 1171

<P>

Parameter Tuning Screen 1646
Parameter Tuning Screen (15/19-inch Display Unit) .1690
Parameter tuning screen (axis setting) 1650
Parameter tuning screen (axis setting) (15/19-inch display unit).....1694
Parameter tuning screen (miscellaneous settings).....1653
Parameter tuning screen (miscellaneous settings) (15/19-inch display unit) 1697
Parameter tuning screen (spindle setting) 1653
Parameter tuning screen (spindle setting) (15/19-inch display unit).....1697
Parameter tuning screen (system setting).....1649
Parameter tuning screen (system setting) (15/19-inch display unit).....1693
PARAMETERS 1827
PART DRAWING AND TOOL MOVEMENT 12
Part Program Storage Size / Number of Registerable Programs 263
Parts Count Display, Run Time Display 936
PASSWORD FUNCTION.....1277
Password modification.....1491
Password modification (15-inch display unit).....1552
Paste.....1284
Path Drawing 1754
PATH GRAPHIC (TOOL POSITION) screen 1765
PATH GRAPHIC screen.....1759
PATH NAME EXPANSION DISPLAY FUNCTION.....1725
PATH TABLE OPERATION.....581
PATTERN DATA INPUT.....542
PC TOOL FOR MEMORY CARD PROGRAM OPERATION/EDITING 2266
Periodic Maintenance Screen.....1658
Periodic Maintenance Screen (15/19-inch Display Unit) 1703
PIVOT AXIS CONTROL.....661
PLANE CONVERSION FUNCTION 171

PLANE SELECTION 170
Plane setting screen 1603
POLAR COORDINATE INTERPOLATION (G12.1, G13.1) 58
POLYGON TURNING (G50.2, G51.2) 609
Position Display in the Relative Coordinate System .. 1346
Position Display in the Workpiece Coordinate System 1344
POSITIONING (G00) 40
Power Consumption Monitoring Screen 1669
Power Consumption Monitoring Screen (15/19-inch Display Unit) 1713
Power Disconnection 938
POWER ON/OFF 937
Precision Level Selection 1496,1499,1559
Precision Level Selection (15/19-inch Display Unit) . 1556
Precision Specification for Macro Relational Operators 454
PREPARATORY FUNCTION (G FUNCTION) 30
Procedure for Reference Position Establishment 998
PROCESSING MACRO STATEMENTS 483
PROGRAM AND FOLDER COPY/MOVE FUNCTION 1315
Program Attributes 260
Program Check Screen 1386
Program Check Screen (15/19-inch Display Unit) 1425
PROGRAM CODE LIST 2149
PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS 266
PROGRAM CONFIGURATION 24,265
Program Contents Display 1372
Program Contents Display (15/19-inch Display Unit) 1416
PROGRAM COPY FUNCTION 1286
Program Display 933
Program Folder Screen 1378
Program Folder Screen (15/19-inch Display Unit) 1419
PROGRAM MANAGEMENT 255,1301
Program Management only in the Path Folder 1322
Program Management under the Path Folder 1321
Program Name 258
PROGRAM RESTART 1031
Program Screen for MDI Operation 1377
Program Screen for MDI Operation (15/19-inch Display Unit) 1419
PROGRAM SEARCH 1272
PROGRAM SECTION CONFIGURATION 268
PROGRAM VERIFICATION 1324
Programmable Data Input (G10) for Blank Figure Drawing Parameters 1776
PROGRAMMABLE MIRROR IMAGE (G50.1, G51.1) 309
PROGRAMMABLE PARAMETER INPUT (G10) 538
PROGRAMS 258
Prohibition of the absolute input by the soft key 1145
Protection Function 1798
Protection level setting 1493
Protection level setting (15-inch display unit) 1553
Protection of Data at Eight Levels 1490
Protection of Data at Eight Levels (15/19-inch Display Unit) 1550
Protection of Various Tool Information Items with the KEY Signal 225

<Q>

QUICK PROGRAM RESTART 1051
Quick Program Restart for a Machining Cycle 1072

<R>

RADIUS DIRECTION ERROR AT CIRCLE CUTTING 2173
RANGE OF COMMAND VALUE 2164
RAPID TRAVERSE 128
RAPID TRAVERSE OVERRIDE 1107
READING AND WRITING VARIABLES FOR ANOTHER PATH 436
READING PARAMETERS 448
Real time macro variables (RTM variables) 511
REAL-TIME CUSTOM MACRO 501
Rectangular parallelepiped setting screen 1601
REFERENCE POSITION 146
Reference Position (Machine-specific Position) 12
REFERENCE POSITION RETURN 146,999
REGISTERING CUSTOM MACRO PROGRAMS 486
Registering, Changing, and Deleting Tool Life Management Data 231
Related Parameters 263
RELATION WITH CONVENTIONAL FUNCTIONS 260
Relation with Folders 260
Relation with Program Names 261
Relationship with Other Functions 1167
Remaining Tool Number Check Function 246
RENAMING A FILE 1313
RENAMING A FOLDER 1309
Repetition (WHILE Statement) 452
Repetition (ZWHILE Statement) 518
Replacement 1283
REPLACING A WORD OR ADDRESS 1269
Replacing batteries 1822
Replacing Battery for Absolute Pulsecoders 1822
Replacing Battery for Control Unit 1818
Replacing the batteries in a separate battery case 1823
Replacing the battery built into the servo amplifier ... 1823
Repositioning 1084
Restriction on machine configurations having table rotation axes (table rotation type and composite type) 892
Restriction on tool rotation type 891
RESTRICTIONS 490,889,1778
Restrictions common to machine configurations 889
RETRACE 1092
Retract 1082
Retract function 658
Return 1083
RETURN FROM THE ALARM SCREEN 1166

Reversing Edit Operations (Undo Function).....1284
RIGID TAPPING BY MANUAL HANDLE.....974
Rotary Axis Control.....618
ROTARY AXIS ROLL-OVER.....617
Rotation axes compensation (G43.4L1, G43.5L1).....744
ROUTINE MAINTENANCE.....1813
RULES OF CHARACTERS IN PROGRAM FILE...2276

<S>

SAFETY FUNCTIONS.....1134
SAFETY PRECAUTIONS.....s-1
Sample programs.....652
Saving.....1285
SCALING (G50, G51).....302
SCHEDULE OPERATION.....1015
SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION 1727
Screen for Assistance in Entering Tilted Working Plane Indexing.....1399
Screen for Assistance in Entering Tilted Working Plane Indexing (15/19-inch Display Unit).....1439
SCREEN HARD COPY FUNCTION.....1255
SCREENS DISPLAYED BY FUNCTION KEY 

.....1344
SCREENS DISPLAYED BY FUNCTION KEY 

.....1372
SCREENS DISPLAYED BY FUNCTION KEY 

.....1457
SCREENS DISPLAYED BY FUNCTION KEY 

.....1627
SCREENS DISPLAYED BY FUNCTION KEY 

.....1715
Search.....1282
SECOND AUXILIARY FUNCTIONS (B CODES) ...252
SELECTING A DEFAULT FOLDER.....1312
SELECTING A DEVICE.....1302
Selecting a From Cassette as a Device.....1306
SELECTING A MAIN PROGRAM.....1315
Selecting a Memory Card Program as a Device.....1302
Selecting a USB Memory as a Device.....1307
Selecting a Workpiece Coordinate System.....160
Selection of a Tool Life Count Period.....225
Selection of machining program.....1804
SELECTION OF TOOL USED FOR VARIOUS MACHINING - TOOL FUNCTION.....22
Sequence Number Comparison and Stop.....1461
Sequence Number Comparison and Stop (15/19-inch Display Unit).....1519
SEQUENCE NUMBER SEARCH.....1273
Servo Parameters.....1630
Servo Parameters (15/19-inch Display Unit).....1674
Servo Tuning.....1631
Servo Tuning (15/19-inch Display Unit).....1675
Setting a Workpiece Coordinate System.....158
Setting an M Code Group Number Using a Program...251
Setting an M Code Group Number Using the Setting Screen.....250
SETTING AND DISPLAY UNITS.....939
SETTING AND DISPLAYING DATA.....1326
Setting and Displaying Data When the Tool Offset for Milling and Turning Function Is Enabled.....1514
Setting and Displaying Data When the Tool Offset for Milling and Turning Function Is Enabled (15/19-inch Display Unit).....1624
Setting and Displaying Tool Management Data.....1472
Setting and Displaying Tool Management Data (15/19-inch Display Unit).....1531
Setting Machine Configuration Data.....906
Setting of Spindle Position / Standby Position Display.....221
Setting Screen.....1151
Setting screens.....1612
Setting the change protection level and output protection level of a program.....1495
Setting the change protection level and output protection level of a program (15-inch display unit).....1555
Setting the Character-codes.....556
Setting the Floating Reference Position.....1355
Setting the Floating Reference Position (15/19-inch Display Unit).....1367
SETTINGS AT POWER-ON, IN THE CLEAR STATE, OR IN THE RESET STATE.....2174
Shape number list screen.....1605
SIMPLE CALCULATION OF INCORRECT THREAD LENGTH.....2168
Simple Call (G65).....457
SIMULTANEOUS EDITING OF MULTIPATH PROGRAMS.....1293
SINGLE BLOCK.....1110
SINGLE DIRECTION POSITIONING (G60).....41
Single-path Display.....1729
SKIP FUNCTION (G31).....115
Skip Function for EGB Axis.....643
SKIP FUNCTION FOR FLEXIBLE SYNCHRONOUS CONTROL.....671
SKIP POSITION MACRO VARIABLE IMPROVEMENT.....118
Small MDI Unit (ONG Key).....943
Smooth control (G43.4P3, G43.5P3).....749
SMOOTH INTERPOLATION (G05.1).....87
Smoothing level selection.....1498,1558
Soft Keys.....949
Special Macro Call using M code.....474
Specification Method.....493
Specification method (G80.5, G81.5).....645
SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5-DIGIT COMMAND).....187
SPECIFYING THE SPINDLE SPEED WITH A CODE.....187
SPINDLE CONTROL BETWEEN EACH PATH.....918
SPINDLE CONTROL WITH SERVO MOTOR.....199
Spindle Electronic Gear Box.....632

Spindle Indexing Function 200
Spindle Monitor 1634
Spindle Monitor (15/19-inch Display Unit) 1678
Spindle Orientation 192
Spindle Positioning 193
SPINDLE POSITIONING FUNCTION 191
Spindle Setting 1632
Spindle Setting (15/19-inch Display Unit) 1676
SPINDLE SPEED FLUCTUATION DETECTION 196
SPINDLE SPEED FUNCTION (S FUNCTION) 187
Spindle Tuning 1633
Spindle Tuning (15/19-inch Display Unit) 1677
SPIRAL INTERPOLATION, CONICAL INTERPOLATION (G02, G03) 52
Split display on the program folder screen 1379,1420
Stamping the Machining Time 1392
Stamping the Machining Time (15/19-inch Display Unit) 1431
Standard MDI Unit (ONG Key) 941
Standard MDI Unit (QWERTY Key) 942
STANDARD PARAMETER SETTING TABLES 2147
Start check signal 1147
STORED STROKE CHECK 1135
STROKE LIMIT AREA CHANGING FUNCTION .1141
STROKE LIMIT CHECK BEFORE MOVE 1139
SUBPROGRAM (M98, M99) 274
Subprogram Call Using a Specific Address 479
Subprogram Call Using an M Code 476
Subprogram Call Using an M Code (Specification of Multiple Definitions) 477
Subprogram Calls Using a Secondary Auxiliary Function 478
Subprogram Calls Using a T Code 477
Subprogram Calls Using an S Code 478
Suppress Motion of Quick Program Restart 1068
SWITCHING BETWEEN MULTI-PATH DISPLAY AND SINGLE-PATH DISPLAY FUNCTION 1718
Switching Machine Configuration 904
Synchronization ratio specification range 655
SYNCHRONOUS, COMPOSITE AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND (G50.4, G51.4, G50.5, G51.5, G50.6, AND G51.6) 614
SYNCHRONOUS/COMPOSITE/SUPERIMPOSED CONTROL 919
System Configuration Screen 1667
System Configuration Screen (15/19-inch Display Unit) 1710
SYSTEM VARIABLES 385,509,513

<T>

Table Horizontal Direction Handle Feed / Table Horizontal Direction JOG Feed / Table Horizontal Direction Incremental Feed 994
Table Vertical Direction Handle Feed / Table Vertical Direction JOG Feed / Table Vertical Direction Incremental Feed 992
TANDEM CONTROL 661
TEMPLATE PROGRAM FUNCTION 1790
TEST OPERATION 1105
TESTING A PROGRAM 928
Text display 1375
Tilted working plane data setting screen 1406,1446
TILTED WORKING PLANE INDEXING 786
Tilted working plane indexing based on Eulerian angle789
Tilted working plane indexing based on projection angles 802
Tilted working plane indexing based on roll-pitch-yaw793
Tilted working plane indexing based on three points ... 795
Tilted working plane indexing based on two vectors ... 799
Tilted working plane indexing by tool axis direction ... 805
Tilted Working Plane Indexing in Tool Length Compensation 840
Tolerance change in High-speed Smooth TCP mode ... 755
Tolerance change in rotation axes compensation (G43.4L1, G43.5L1) 755
Tolerance change in smooth control (G43.4P3, G43.5P3) 758
Tool Axis Direction Control 819
Tool Axis Direction Handle Feed / Tool Axis Direction JOG Feed / Tool Axis Direction Incremental Feed .. 985
Tool Axis Right-Angle Direction Handle Feed / Tool Axis Right-Angle Direction JOG Feed / Tool Axis Right-Angle Direction Incremental Feed 987
TOOL CENTER POINT CONTROL 682
Tool center point retention type tool axis direction control 836
TOOL FUNCTION (T FUNCTION) 202
Tool geometry data screen 1484
Tool geometry data screen (15-inch display unit) 1544
Tool holder and object monitor screen 1580
Tool holder figure setting screen 1585
TOOL LENGTH COMPENSATION (G43, G44, G49) 295
Tool Life Count Restart M Code 244
Tool Life Counting and Tool Selection 242
TOOL LIFE MANAGEMENT 228
Tool life management (group editing screen) 1505
Tool life management (group editing screen) (15-inch display unit) 1565
Tool life management (list screen) 1502
Tool life management (list screen) (15-inch display unit) 1562
Tool Life Management Commands in Machining Program 236
Tool Life Management Data 229
TOOL MANAGEMENT EXTENSION FUNCTION .217
TOOL MANAGEMENT FUNCTION 203
TOOL MANAGEMENT FUNCTION FOR OVERSIZE TOOLS 226
Tool monitor screen 1579
TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE-INTERPOLATION 9
TOOL MOVEMENT BY PROGRAMING - AUTOMATIC OPERATION 926
TOOL MOVEMENT RANGE - STROKE 26

TOOL OFFSET CONVERSION FUNCTION (G44.1)367
TOOL OFFSET FOR MILLING AND TURNING FUNCTION (G43.7) 361
Tool offset range setting screen 1153
TOOL PATH AT CORNER 2170
TOOL POSTURE CONTROL.....766
TOOL RETRACT AND RECOVER.....619,1080
Tool Retract and Recover for Threading.....1084
TOOL SELECTION FUNCTION 202
Tool side offset.....853
Tool Tip Center Rotation Handle Feed / Tool Tip Center Rotation JOG Feed / Tool Tip Center Rotation Incremental Feed 990
Tool tip position (cutting point) command.....873
TORQUE LIMIT SKIP.....119
Total Life Time Display for Tools of The Same Type.226
Turning on the Power.....937
Two-path Display and Three-path Display 1730
TYPES OF REAL TIME MACRO COMMANDS.....503

<U>

U-axis Control.....658
U-axis Control 2 Pairs.....660
Unconditional Branch (GOTO Statement).....450
Usable Characters in Program File.....2276
Usage Notes 2266
USB FUNCTION.....1256

<V>

VARIABLE LEAD THREADING (G34).....110
VARIABLES 380,508
Variables Dedicated to Real Time Custom Macros 509
VIRTUAL MDI KEY 1786
VIRTUAL MDI KEY FUNCTION 1786

<W>

WAITING FUNCTION FOR PATHS 912
Warning Messages 1777
WARNINGS AND CAUTIONS RELATED TO HANDLING.....s-5
WARNINGS AND CAUTIONS RELATED TO PROGRAMMING.....s-3
WARNINGS RELATED TO DAILY MAINTENANCE.....s-7
Withdrawal.....1083
Word Search.....1265
WORKPIECE COORDINATE SYSTEM 158
Workpiece Coordinate System Preset 1350
Workpiece Coordinate System Preset (15/19-inch Display Unit) 1363
Workpiece Coordinate System Preset (G92.1).....163
Workpiece Coordinate System Shift 167
Workpiece origin offset range setting screen 1156
WORKPIECE SETTING ERROR COMPENSATION.....315
Workpiece shift range setting screen.....1159
WRONG OPERATION PREVENTION FUNCTIONS.....1142

<Y>

Y-axis tool offset range setting screen 1158

REVISION RECORD

Edition	Date	Contents
03	Aug., 2011	• Addition of following items - Tool offset conversion function (G44.1) - Macro call argument for axis name expansion - Chopping function - Skip function for flexible synchronous control - Chopping function by flexible synchronous control - High precision oscillation function - Displaying program being executed in backward movement - Inputting and outputting of O8-digit - Inputting/outputting all programs and folders - Changing current folder - Program verification - Folder tree display - External operator message history • Correction of descriptions of following items - Tilted working plane indexing - Program and folder copy/move function • Deletion of following items - Jerk control - Optimum torque acceleration/deceleration - Arbitrary angular axis control - Making a program compact - Displaying and setting pitch error compensation data - Displaying and setting 3-dimensional error compensation data - Displaying and setting three-dimensional rotary error compensation data • Correction of errors
02	Oct., 2010	• Addition of following items - Plane conversion function - Special macro call using M code - Improvement of tool compensation for tool retract and recover - High-speed smooth TCP - Machine configuration selecting function - Inputting and outputting three-dimensional rotary error compensation data - Folder management - Displaying and setting three-dimensional rotary error compensation data - Power consumption monitoring screen - Multi-path program management function - ISO/ASCII code conversion tool • Correction of errors
01	Jun., 2010	

ADDITIONAL INFORMATION

FANUC Series 30*i*/31*i*/32*i*-MODEL A
FANUC Series 30*i*/31*i*/32*i*/35*i*-MODEL B
Display function for Display unit for Automotive

1. Type of applied technical documents

Name	FANUC Series 30*i*–MODEL A FANUC Series 31*i*–MODEL A FANUC Series 32*i*–MODEL A Common to Lathe System / Machining Center System OPERATOR'S MANUAL
Spec.No./Version	B-63944EN/04

Name	FANUC Series 30*i*–MODEL B FANUC Series 31*i*–MODEL B FANUC Series 32*i*–MODEL B Common to Lathe System / Machining Center System OPERATOR'S MANUAL
Spec.No./Version	B-64484EN/02

Name	FANUC Series 35*i*–MODEL B OPERATOR'S MANUAL
Spec.No./Version	B-64524EN/01

2. Summary of change

Group	Name / Outline	New, Add, Correct, Delete	Applicable Date
Basic Function			
Optional Function	Display function for Display unit for Automotive	Add	Immediately
Unit			
Maintenance parts			
Notice			
Correction			
Another			

Add the following description as **III**.OPERATION 2.1.9 “Display function for Display unit for Automotive” in FANUC Series 30*i*/31*i*/32*i*-A Common to Lathe System / Machining Center System OPERATOR’S MANUAL (B-63944EN).
Add the following description as **III**.OPERATION 2.2.6 “Display function for Display unit for Automotive” in FANUC Series 30*i*/31*i*/32*i*-B Common to Lathe System / Machining Center System OPERATOR’S MANUAL (B-64484EN).
Add the following description as **III**.OPERATION 2.2.6 “Display function for Display unit for Automotive” in FANUC Series 35*i*-B OPERATOR’S MANUAL (B-64524EN).

2.2.6 Display function for Display unit for Automotive

（(In case FANUC Series 30*i*/31*i*/32*i*-A Common to Lathe System / Machining Center System OPERATOR’S MANUAL (B-63944EN), the chapter number is 2.1.9）

Overview

By using this function, the soft key is displayed to the position of the device key of Display unit for Automotive and Panel *i* for Automotive.



Fig. 2.2.6 (a) Display unit for Automotive

Vertical soft key

In the case of Display unit for Automotive or Panel *i* for Automotive, the 9th soft key of the vertical soft key is the [F13] key. When the [F13] key is pressed, one touch menu key is displayed.

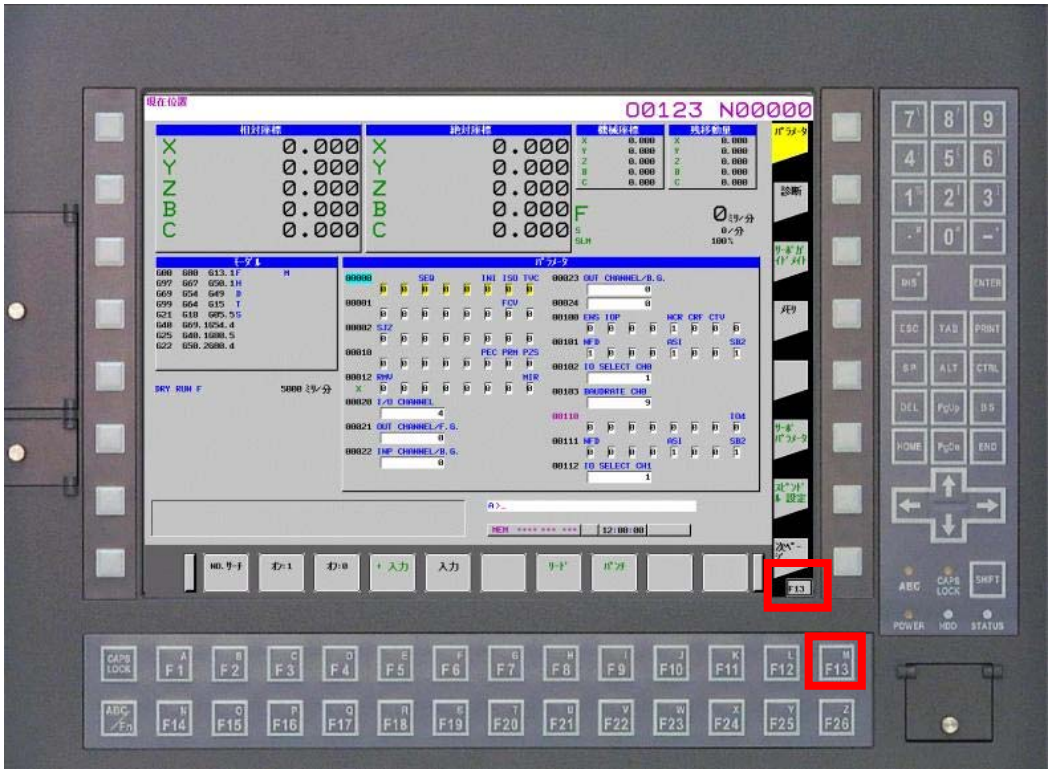


Fig. 2.2.6 (b) Vertical soft key [F13]

Function key

There is no function key and HELP key in the Display unit for Automotive and Panel *i* for Automotive.
In the case of Display unit for Automotive, the operation of these keys is done by the following key operation.

Function key	
POS	CTRL+F2
PROG	CTRL+F3
OFFSET	CTRL+F4
SYSTEM	CTRL+F5
MESSAGE	CTRL+F6
GRAPH	CTRL+F7
CUSTOM1	CTRL+F8
CUSTOM2	CTRL+F9
HELP	ESC

In the case of Panel *i* for Automotive, these key operation is set by keyboard assignment. (For details, refer to chapter 3.6 "CHANGING KEYBOARD ASSIGNMENT" of CNC Screen Display Function OPERATOR'S MANUAL(B-63164EN).)
In the initial state, the operation of these keys is done by the following key operation.

Function key	
POS	CTRL+F1
PROG	CTRL+F2
OFFSET	CTRL+F3
SYSTEM	CTRL+F4
MESSAGE	CTRL+F5
GRAPH	CTRL+F6
CUSTOM1	CTRL+F7
CUSTOM2	CTRL+F8
HELP	ESC

Above keys can be assigned to the one touch menu.
After assignment, function key and HELP key can input from the one touch menu.

> **⚠ CAUTION**
> In the case of display unit with I/O Link adaptor, output of signal to PMC is done with regardless of the kind of the screen by pressing the vertical soft key. So, it is necessary to pay attention to miss-operation.

> **NOTE**
> As the one touch menu is not effective on the screen of C Language executor and macro executor, it is necessary to use the CTRL+F2-F9 to change to the CNC screen.

To assign the function key to the one touch menu, it is necessary to set the screen number to the parameter No.3321 to 3336.
The screen number is described in the following table.

Table. 2.2.6 (a) Function key display to one touch menu

Screen number	Function key	
1	POSITION	
2	PROGRAM	
3	OFFSET/ SETTING	
4	SYSTEM	
5	MESSAGE	

Screen number	Function key	
6	GRAPH	
7	CUSTOM 1	
8	CUSTOM 2	
9	HELP	

The information of other screen numbers that can be set to one touch menu is described in FANUC Series 30*i*/31*i*/32*i*-B PARAMETER MANUAL (B-64490EN) (In case FANUC Series 30*i*/31*i*/32*i*-A, the PARAMETER MANUAL is B-63950EN. And in case FANUC Series 35*i*-B, the PARAMETER MANUAL is B-64530EN.)

PARAMETER

	#7	#6	#5	#4	#3	#2	#1	#0
11355							CDA	

[Input type] Parameter input
[Data type] Bit

#1 CDA The 15 inch display unit is
0: Regular type

1: Display unit for Automotive

	#7	#6	#5	#4	#3	#2	#1	#0
3208					NOS			

[Input type] Setting input
[Data type] Bit

#3 NOS One touch menu is
0: displayed
1: not displayed

3321	Screen number assigned to the 1st vertical soft key
	～
3336	Screen number assigned to the 16th vertical soft key

[Input type] Parameter input
[Data type] Word
[Valid data range] 1 to 10000
Assign a screen number to be displayed as a shortcut to a vertical soft key.
The 1st to 8th vertical soft keys are displayed on page 1, and the 9th to 16th vertical soft keys are displayed on page 2.
When specifying page 2, be sure to specify "Display of next page" (Set the number to "99") on each page.
When not specifying page 2, set 0 for the 9th to 16th soft keys. In this case, page 2 is not used, so that "Display of next page" need not be specified on page 1.