FANUC Series 0*i*-MODEL D
FANUC Series 0*i* Mate-MODEL D

CONNECTION MANUAL (FUNCTION)

(Volume 1 of 2)

B-64303EN-1/02

- 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 from Japan may be subject to an export license by the government of Japan.
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 using any of the products in a facility in which a serious accident or loss is predicted due to a failure of the product, install a safety device.

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".

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.

PREFACE

Organization of this manual

This manual describes all the NC functions required to enable machine tool builders to design their CNC machine tools. The following items are explained for each function.

1. Overview
Describes feature of the function. Refer to Operator's Manual as requied.

2. Signal
Describes names, functions, output conditions and addresses of the signals required to realize a function.

3. Parameter
Describes parameters related with a function.

4. Alarms and message
Lists the alarms and messages related with a function in a table.

5. Reference item
List the related items of the related manuals in a table.

A list of addresses of all signals and a list of signals are described in the appendix of this manual. Refer to it as required.

Applicable models

This manual describes the following models that are 'Nano CNC'.
'Nano CNC system' which realizes high precision machining can be constructed by combining these models and high speed, high precision servo controls.
In the text, the abbreviations may be used in addition to Model name indicated below.

Model name	Abbreviation		
FANUC Series 0*i* -TD	0*i* -TD	Series 0*i* -D	0*i* -D
FANUC Series 0*i* -MD	0*i* -MD		
FANUC Series 0*i* Mate -TD	0*i* Mate -TD	Series 0*i* Mate -D	0*i* Mate -D
FANUC Series 0*i* Mate -MD	0*i* Mate -MD		

NOTE
1 For an explanatory purpose, the following descriptions may be used according to the types of path control used:
- T series: 0*i* -TD / 0*i* Mate -TD
- M series: 0*i* -MD / 0*i* Mate -MD

2 Some functions described in this manual may not be applied to some products. For details, refer to the DESCRIPTIONS (B-64302EN).

Description of symbols

The following symbols are used in this manual. These symbols are described below.

- **M**

Indicates a description that is valid only for the M series (machining center system).
In a general description of the method of machining, an M series operation is identified by a phase such as "for milling machining".

- **T**

Indicates a description that is valid only for the T series (lathe system).
In a general description of the method of machining, a T series operation is identified by a phrase such as "for lathe cutting".

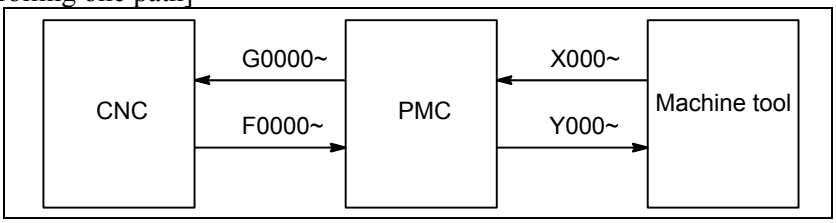
-

Indicates the end of a description of a control type.
When a control type mark mentioned above is not followed by this mark, the description of the 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.

Description of signals

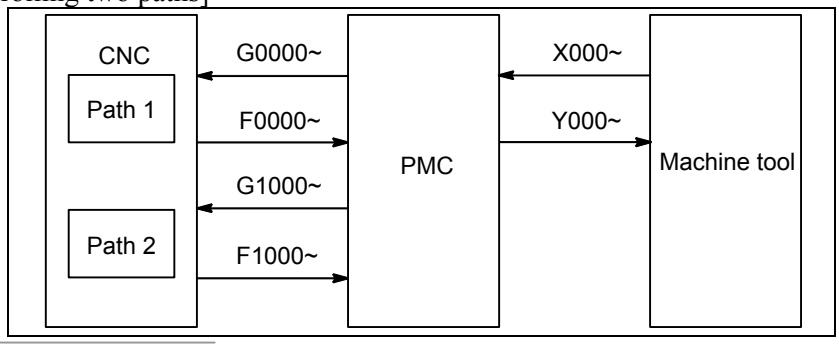
Relation of interface signals among the CNC, the PMC and the machine tool is shown below:

[Example of controlling one path]



T

[Example of controlling two paths]



- Expression of signals

Address	Symbol (#0 to #7 indicates bit position)							
	#7	#6	#5	#4	#3	#2	#1	#0
Fn000	OP	SA	STL	SPL				RWD

In an item where both T series and M series are described, some signals are covered with shade ( ) in the signal address figure as shown below. This means either lathe system or machining center system does not have this signal. Upper part is for lathe system and lower part is for machining center system.

	#7	#6	#5	#4	#3	#2	#1	#0	
Gn053	*CDZ		ROVLP		UINT			TMRON	T series M series

[Example 1]

The figure above indicates *CDZ is provided only for the T series while the other signals for both the T series and M series.

	#7	#6	#5	#4	#3	#2	#1	#0	
Gn040					OFN9	OFN8	OFN7	OFN6	T series M series

[Example 2]

Signals OFN6 to OFN9 are for M series only.

NOTE

1 The following notational conventions are used in the signal description of each function.

Example) Axis moving signals MV1 to MV5 <Fn102.0 to Fn102.4>

↑ ↑ ↑

Signal name Symbol name Signal address

2 One of the following superscripts is attached to the top right of a symbol depending on the signal type.
- Path type (on CNC side) : #P
- Controlled axis type (on CNC side) : #SV
- Spindle type (on CNC side) : #SP
- PMC axis control group type : #PX

Refer to Appendix “List of Addresses“ for details

3 For the signals, a single data number is assigned to 8 bits. Each bit has a different meaning.

4 The letter "n" in each address representation indicates the address position used in each path on the CNC side, as shown below.

1st path : n=0 (No. 0 to 999)

2nd path : n=1 (No. 1000 to 1999)

Description of parameters

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. For the 0*i* -D/0*i* Mate-D, the maximum number of machine groups is always 1.
3 For path types, parameters for two 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.

- 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.

(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
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

(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
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

(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	0.0 to +999000.00
	IS-B	0.001	0.0 to +999000.000
	IS-C	0.0001	0.0 to +99999.9999
inch/min	IS-A	0.001	0.0 to +96000.000
	IS-B	0.0001	0.0 to +9600.0000
	IS-C	0.00001	0.0 to +4000.00000

If bit 7 (IESP) of parameter No. 1013 is set to 1, the valid data ranges for IS-C are extended as follows:

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min degree/min	IS-C	0.001	0.000 to +999000.000
inch/min	IS-C	0.0001	0.0000 to +9600.0000

(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
$inch/sec^2$	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

If bit 7 (IESP) of parameter No. 1013 is set to 1, the valid data ranges for IS-C are extended as follows:

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min degree/min	IS-C	0.001	0.000 to +999999.999
inch/min	IS-C	0.0001	0.0000 to +99999.9999

⚠ CAUTION
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" of the Operator's Manual (B-64304EN).

- **Parameters of the bit type, bit machine group type, bit path type, bit axis type, and bit spindle type**

Data No.	Data (Data #0 to #7 are bit positions.)							
	#7	#6	#5	#4	#3	#2	#1	#0
0000			SEQ			INI	ISO	TVC

- **Parameters other than the parameters above**

Data No.	Data
1023	Number of the servo axis for each axis

NOTE
1 The bits left blank in "description of parameters" and parameter numbers that appear on the display but are not found in the parameter list are reserved for future expansion. Usually, set these parameters to 0.

NOTE

2 Each parameter is indicated in either or both of two stages according to whether it is valid for the M series, T series, or both, as illustrated by the following examples. When a row is blank, the parameter is not usable with the corresponding series.

[Example 1]

Parameter HTG is a parameter common to the M and T series, but Parameters RTV and ROC are parameters valid only for the T series.

	#7	#6	#5	#4	#3	#2	#1	#0	
1403	RTV		HTG	ROC					T series
			HTG						M series

[Example 2]

The following parameter is provided only for the M series.

1411		T series
	Cutting feedrate	M series

3 When "to" is inserted between two parameter numbers, there are parameters with successive numbers between the two starting and ending parameter numbers, but those intermediate parameter numbers are omitted for convenience.

4 The lower-case letter "x" or "s" following the name of a bit-type parameter indicates the following:

- "□□□x" : Bit axis type parameters
- "◯◯◯s" : Bit spindle type parameters

Related manuals of Series 0*i* -D,Series 0*i* Mate -D

The following table lists the manuals related to Series 0*i* -D,Series 0*i* Mate -D. This manual is indicated by an asterisk(*).

Table 1 Related manuals

Manual name	Specification number	
DESCRIPTIONS	B-64302EN	
CONNECTION MANUAL (HARDWARE)	B-64303EN	
CONNECTION MANUAL (FUNCTION)	B-64303EN-1	*
OPERATOR'S MANUAL (Common to Lathe System/Machining Center System)	B-64304EN	
OPERATOR'S MANUAL (For Lathe System)	B-64304EN-1	
OPERATOR'S MANUAL (For Machining Center System)	B-64304EN-2	
MAINTENANCE MANUAL	B-64305EN	
PARAMETER MANUAL	B-64310EN	
START-UP MANUAL	B-64304EN-3	
Programming		
Macro Executor PROGRAMMING MANUAL	B-64303EN-2	
Macro Compiler PROGRAMMING MANUAL	B-64303EN-5	
C Language Executor PROGRAMMING MANUAL	B-64303EN-3	
PMC		
PMC PROGRAMMING MANUAL	B-64393EN	
Network		
PROFIBUS-DP Board CONNECTION MANUAL	B-64403EN	
Fast Ethernet / Fast Data Server OPERATOR'S MANUAL	B-64414EN	
DeviceNet Board CONNECTION MANUAL	B-64443EN	
FL-net Board CONNECTION MANUAL	B-64453EN	

Manual name	Specification number	
Dual Check Safety		
Dual Check Safety CONNECTION MANUAL	B-64303EN-4	
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	
MANUAL GUIDE 0*i* OPERATOR'S MANUAL	B-64434EN	
TURN MATE *i* OPERATOR'S MANUAL	B-64254EN	

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*/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

This manual mainly assumes that the FANUC SERVO MOTOR α*i* 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.

TABLE OF CONTENTS

Volume 1 of 2

DEFINITION OF WARNING, CAUTION, AND NOTE s-1

PREFACE p-1

1 AXIS CONTROL 1
1.1 CONTROLLED AXIS 1
1.2 SETTING EACH AXIS 2
1.2.1 Name of Axes 2
1.2.2 Increment System 4
1.2.3 Specifying the Rotation Axis 7
1.2.4 Controlled Axes Detach 10
1.2.5 Outputting the Movement State of an Axis 13
1.2.6 Mirror Image 14
1.2.7 Follow-up 16
1.2.8 Servo off/Mechanical Handle Feed 17
1.2.9 Position Switch 19
1.3 ERROR COMPENSATION 22
1.3.1 Stored Pitch Error Compensation 22
1.3.2 Backlash Compensation 28
1.3.3 Smooth Backlash 30
1.3.4 Simple Straightness Compensation (M Series) 33
1.3.5 Gradient Compensation 37
1.3.6 Bi-directional Pitch Error Compensation 40
1.3.7 Differences between Pitch Error Compensation, Simple Straightness Compensation, and Gradient Compensation (Reference) 47
1.4 SETTINGS RELATED TO SERVO-CONTROLLED AXES 48
1.4.1 Parameters Related to Servo 48
1.4.2 Absolute Position Detection 54
1.4.3 FSSB Setting 60
1.4.3.1 Series 0*i*-D-dedicated setting 60
1.4.3.2 Series 0*i*-C-compatible setting 76
1.4.4 Temporary Absolute Coordinate Setting 89
1.5 SETTINGS RELATED WITH COORDINATE SYSTEMS 92
1.5.1 Machine Coordinate System 92
1.5.2 Workpiece Coordinate System/Addition of Workpiece Coordinate System Pair 94
1.5.2.1 Workpiece coordinate system 94
1.5.2.2 Workpiece coordinate system preset 97
1.5.2.3 Adding workpiece coordinate systems (G54.1 or G54) (M series) 98
1.5.2.4 Automatic coordinate system setting 99
1.5.2.5 Workpiece coordinate system shift (T series) 100
1.5.2.6 Each axis workpiece coordinate system preset signals 104
1.5.3 Local Coordinate System 109
1.5.4 Rotary Axis Roll Over 111
1.6 AXIS SYNCHRONOUS CONTROL 113
1.6.1 Axis Configuration for Axis Synchronous Control 113
1.6.2 Synchronization Establishment 116
1.6.3 Automatic Setting for Grid Position Matching 118
1.6.4 Synchronization Error Check 118

1.6.5 Methods of Alarm Recovery by Synchronization Error Check 119
1.6.6 Axis Synchronous Control Torque Difference Alarm....120
1.6.7 Automatic Slave Axis Parameter Setting 121
1.7 TANDEM CONTROL 136
1.8 ARBITRARY ANGULAR AXIS CONTROL 144
1.9 ELECTRONIC GEAR BOX (M SERIES) 156
1.9.1 Electronic Gear Box (M Series) 156

2 PREPARATIONS FOR OPERATION 177
2.1 EMERGENCY STOP 177
2.2 CNC READY SIGNALS 179
2.3 OVERTRAVEL CHECK 180
2.3.1 Overtravel Signals 180
2.3.2 Stored Stroke Check 1 182
2.3.3 Stored Stroke Check 2, 3 187
2.3.4 Checking the Stored Stroke Limit during the Time from Power–on to the Reference Position Establishment 194
2.3.5 Stroke Limit External Setting (M Series) 196
2.3.6 Chuck and Tail Stock Barrier (T Series) 197
2.3.7 Stroke Limit Check Before Move 206
2.4 ALARM SIGNALS 209
2.5 START LOCK / INTERLOCK 210
2.6 MODE SELECTION 216
2.7 STATUS OUTPUT SIGNAL 222
2.8 VRDY OFF ALARM IGNORE SIGNAL 223
2.9 UNEXPECTED DISTURBANCE TORQUE DETECTION FUNCTION 225
2.10 MACHINING CONDITION SELECTION FUNCTION 235
2.11 MACHINING QUALITY LEVEL ADJUSTMENT (M Series) 241
2.12 MALFUNCTION PREVENT FUNCTIONS 243
2.13 OPERATOR ERROR PREVENT FUNCTIONS 246

3 MANUAL OPERATION 256
3.1 JOG FEED/INCREMENTAL FEED 256
3.2 MANUAL HANDLE FEED 262
3.3 MANUAL HANDLE INTERRUPT 271
3.4 I/O Link β MANUAL HANDLE INTERFACE 276
3.5 MANUAL HANDLE RETRACE 279

4 REFERENCE POSITION ESTABLISHMENT 299
4.1 MANUAL REFERENCE POSITION RETURN 299
4.2 REFERENCE POSITION SETTING WITHOUT DOG 320
4.3 AUTOMATIC REFERENCE POSITION RETURN AND RETURN FROM THE REFERENCE POSITION 327
4.4 2ND REFERENCE POSITION RETURN / 3RD, 4TH REFERENCE POSITION RETURN 332
4.5 REFERENCE POSITION SETTING WITH MECHANICAL STOPPER 334
4.6 DISTANCE CODED LINEAR SCALE INTERFACE 339
4.7 LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL) 351

4.8 EXTENDED FUNCTION OF THE DISTANCE CODED LINEAR SCALE INTERFACE 361
4.8.1 Reference Position Established by the G00 Command 361
4.8.2 Reference Position Establishment by Jog Feed 365
4.9 CORRESPONDENCE OF ROTARY SCALE WITHOUT ROTARY DATA. 368
4.9.1 Setting Method by Rotary Axis Type and Movable Range 368
4.9.2 In the Case of a Rotary Axis B Type whose Movable Range is under One Rotation 369
4.9.3 In the Case of a Rotary Axis B Type whose Movable Range is over One Rotation 375
4.9.4 In the Case of a Rotary Axis A Type 377
4.9.5 Method of Using Heidenhain Rotary Scale RCN223, 723 and 220 378

5 AUTOMATIC OPERATION 381
5.1 CYCLE START/FEED HOLD 381
5.2 RESET AND REWIND 384
5.3 TESTING A PROGRAM 388
5.3.1 Machine Lock 388
5.3.2 Dry Run 389
5.3.3 Single Block 391
5.4 MANUAL ABSOLUTE ON/OFF 393
5.5 OPTIONAL BLOCK SKIP/ADDITION OF OPTIONAL BLOCK SKIP 395
5.6 SEQUENCE NUMBER COMPARISON AND STOP 397
5.7 PROGRAM RESTART 397
5.8 RETRACE (M SERIES) 403
5.9 EXACT STOP / EXACT STOP MODE / TAPPING MODE / CUTTING MODE 413
5.10 RETRACTION FOR RIGID TAPPING 414
5.11 DNC OPERATION 419
5.12 MANUAL INTERVENTION AND RETURN 422
5.13 DIRECT OPERATION BY C LANGUAGE EXECUTOR 425

6 INTERPOLATION FUNCTION 426
6.1 POSITIONING 426
6.2 SINGLE DIRECTION POSITIONING (M SERIES) 427
6.3 LINEAR INTERPOLATION 430
6.4 CIRCULAR INTERPOLATION 432
6.5 THREADING 435
6.5.1 Threading 435
6.5.2 Threading Cycle Retract (Canned Cycle) (T Series) 440
6.5.3 Threading Cycle Retract (Multiple Repetitive Canned Cycle) (T Series) 443
6.5.4 Variable Lead Threading (T Series) 447
6.5.5 Continuous Threading (T Series) 447
6.6 HELICAL INTERPOLATION 448
6.7 POLAR COORDINATE INTERPOLATION (T SERIES) 449
6.8 CYLINDRICAL INTERPOLATION 452
6.8.1 Cylindrical Interpolation 452
6.9 POLYGON TURNING (T SERIES) 454
6.9.1 Polygon Turning 455
6.9.2 Polygon Turning with Two Spindles 462

6.10 NORMAL DIRECTION CONTROL (M SERIES) 480
6.11 LINEAR INTERPOLATION (G28, G30, G53) 483
6.12 NANO SMOOTHING (M SERIES) 484
6.13 GENERAL PURPOSE RETRACT 491

7 FEEDRATE CONTROL/ACCELERATION AND DECELERATION CONTROL 496

7.1 FEEDRATE CONTROL 496
7.1.1 Rapid Traverse Rate 497
7.1.2 Cutting Feedrate Clamp 499
7.1.3 Feed Per Minute 499
7.1.4 Feed Per Revolution/Manual Feed Per Revolution 501
7.1.5 One-digit F Code Feed (M Series) 502
7.1.6 Inverse Time Feed (M Series) 505
7.1.7 Override 505
7.1.7.1 Rapid traverse override 505
7.1.7.2 Feedrate override 509
7.1.7.3 Override cancel 511
7.1.8 Automatic Corner Override (M Series) 511
7.1.8.1 Inner corner automatic override (G62) 512
7.1.8.2 Internal circular cutting feedrate change 514
7.1.9 External Deceleration 516
7.1.10 Acceleration-Based Speed Control in Circular Interpolation 521
7.1.11 High Precision and High Speed Functions (Advanced Preview Control (T Series) / AI Advanced Preview Control (M Series) / AI Contour Control (M Series) / AI Contour Control II (M Series)) 522
7.1.12 Speed Command Extension in Least Input Increment C 552
7.2 ACCELERATION/DECELERATION CONTROL 555
7.2.1 Automatic Acceleration/Deceleration 555
7.2.1.1 Automatic acceleration/deceleration 555
7.2.1.2 Rapid traverse block overlap 559
7.2.2 Rapid Traverse Bell-shaped Acceleration/Deceleration 560
7.2.3 Linear Acceleration/Deceleration after Cutting Feed Interpolation 562
7.2.4 Bell-Shaped Acceleration/Deceleration after Cutting Feed Interpolation (M Series) 565
7.2.5 Corner Control 567
7.2.5.1 In-position check signal 567
7.2.5.2 In-position check 568
7.2.5.3 In-position check independently of feed/rapid traverse 570
7.2.6 Feed Forward in Rapid Traverse 572
7.3 JERK CONTROL (M Series) 572
7.3.1 Speed Control with Change of Acceleration on Each Axis 572
7.3.2 Look-Ahead Smooth Bell-Shaped Acceleration/Deceleration before Interpolation 576

8 2-PATH CONTROL 578

8.1 2-PATH CONTROL 578
8.1.1 CNC Data Display, Setup, and Input/Output 583
8.1.2 2-path Functions 583
8.1.3 Cautions on 2-path Control 585
8.2 WAITING M CODES 591
8.3 PATH INTERFERENCE CHECK 593
8.4 BALANCE CUTTING 602

8.5 SYNCHRONOUS CONTROL AND COMPOSITE CONTROL 605
8.5.1 Synchronous Control 607
8.5.2 Composite Control 613
8.5.3 Hypothetical Cs Axis Control 646
8.6 SUPERIMPOSED CONTROL 650
8.7 SYNCHRONOUS, COMPOSITE, AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND 663
8.8 PATH SPINDLE CONTROL 665
8.9 MEMORY COMMON TO PATHS 675
8.10 PATH SINGLE BLOCK CHECK FUNCTION 678
8.11 PATH SELECTION/DISPLAY OF OPTIONAL PATH NAMES 679

9 AUXILIARY FUNCTION 682
9.1 AUXILIARY FUNCTION/2ND AUXILIARY FUNCTION 682
9.2 AUXILIARY FUNCTION LOCK 694
9.3 MULTIPLE M COMMANDS IN A SINGLE BLOCK 695
9.4 HIGH-SPEED M/S/T/B INTERFACE 697

10 SPINDLE SPEED FUNCTION 700
10.1 SPINDLE SPEED FUNCTION (S CODE OUTPUT) 700
10.2 DESIGNATION OF SPINDLE AXES 701
10.3 SPINDLE SERIAL OUTPUT 705
10.4 SPINDLE ANALOG OUTPUT 715
10.5 SERIAL/ANALOG SPINDLE CONTROL 719
10.6 SPINDLE SPEED CONTROL 722
10.7 SPINDLE OUTPUT CONTROL BY THE PMC 749
10.8 CONSTANT SURFACE SPEED CONTROL 755
10.9 ACTUAL SPINDLE SPEED OUTPUT (T SERIES) 763
10.10 SPINDLE POSITIONING (T SERIES) 764
10.11 Cs CONTOUR CONTROL 797
10.11.1 Cs Contour Control 797
10.11.2 Cs Contour Control Torque Limit Skip 815
10.11.3 Cs Contour Control Axis Coordinate Establishment 818
10.12 MULTI-SPINDLE CONTROL 826
10.13 RIGID TAPPING 843
10.13.1 Connection Among Spindle, Spindle Motor, and Position Coder 844
10.13.2 Rigid Tapping Specification 848
10.13.3 Commands for Feed Per Minute and Feed Per Revolution 849
10.13.4 Acceleration/Deceleration after Interpolation 849
10.13.5 Override 851
10.13.6 Reference Position Return 853
10.13.7 FS10/11 Format Command 854
10.13.8 Multi Spindle Control 856
10.13.9 Rigid Tapping with Spindle of Another Path (T Series (2-ptah Control)) 856
10.13.10 Optimum Acceleration/Deceleration for Rigid Tapping 860
10.13.11 Notes 866
10.13.12 Diagnosis Display 867
10.13.13 Command Format 870
10.13.14 Position Control Loop Gain Parameter Switching 874
10.13.15 Signal 875
10.13.15.1 Signals for the rigid tapping function 875

10.13.15.2 Signals related to S code output 875
10.13.15.3 Signals related to gear switching 876
10.13.15.4 Signals related to the addition of multi spindle control 877
10.13.15.5 Notes on interface with the PMC 880
10.13.16 Timing Charts for Rigid Tapping Specification 882
10.13.16.1 When M29 is specified before G84/G74 883
10.13.16.2 M29 and G84/G74 are specified in the same block 887
10.13.16.3 Specifying G84/G74 for rigid tapping by parameters 891
10.13.16.4 When M29 is specified before G84/G88 895
10.13.16.5 M29 and G84/G88 are specified in the same block 897
10.13.16.6 Specifying G84/G88 for rigid tapping by parameters 899
10.13.16.7 Timing of the M code for unclamping (T series) 901
10.13.16.8 Timing to cancel rigid tapping mode 901
10.13.17 Parameter 902
10.13.18 Notes 923
10.14 SPINDLE SYNCHRONOUS CONTROL 927
10.15 SPINDLE ORIENTATION 945
10.16 SPINDLE OUTPUT SWITCHING 948
10.17 SIMPLE SPINDLE SYNCHRONOUS CONTROL (M SERIES) 949
10.18 SPINDLE SPEED FLUCTUATION DETECTION (T SERIES) 960
10.19 SPINDLE CONTROL WITH SERVO MOTOR 970
10.19.1 Spindle Control with Servo Motor 972
10.19.2 Spindle Indexing Function 996
10.19.3 Rigid Tapping with Servo Motor 1002
10.19.4 Feed per Revolution 1007
10.19.5 Spindle Output Control with PMC 1008

Volume 2 of 2

11 TOOL FUNCTIONS 1009
11.1 TOOL FUNCTIONS OF T SERIES 1009
11.1.1 Tool Offset 1010
11.1.2 Tool Geometry Offset and Tool Wear Offset 1010
11.1.3 Offset 1011
11.1.4 Automatic Alteration of Tool Position Compensation (T Function) 1017
11.2 TOOL FUNCTIONS OF M SERIES 1021
11.2.1 Tool Compensation Memory 1022
11.3 TOOL COMPENSATION 1027
11.3.1 Cutter Compensation (M Series) and Tool Nose Radius Compensation (T Series) 1027
11.3.2 Tool Length Compensation 1035
11.3.3 Tool Length Compensation Shift Types 1038
11.4 TOOL LIFE MANAGEMENT 1042

12 PROGRAM COMMAND 1059
12.1 DECIMAL POINT PROGRAMMING/POCKET CALCULATOR TYPE DECIMAL POINT PROGRAMMING 1059
12.2 G CODE SYSTEM 1061
12.2.1 G Code List in the Lathe System 1061
12.2.2 G Code List in the Machining Center System 1063
12.3 PROGRAM CONFIGURATION 1067

12.4 PART PROGRAM STORAGE SIZE / NUMBER OF REGISTERABLE PROGRAMS 1069
12.5 INCH/METRIC CONVERSION 1070
12.6 CUSTOM MACRO 1075
12.6.1 Custom Macro 1075
12.6.2 Interruption Type Custom Macro 1093
12.7 CANNED CYCLE FOR DRILLING 1096
12.8 CANNED CYCLE (T SERIES) / MULTIPLE REPETITIVE CANNED CYCLE (T SERIES) 1111
12.9 IN-FEED CONTROL (FOR GRINDING MACHINE) (M SERIES) 1120
12.10 CANNED GRINDING CYCLE (FOR GRINDING MACHINE) 1121
12.11 MIRROR IMAGE FOR DOUBLE TURRET (T SERIES) 1126
12.12 INDEX TABLE INDEXING (M SERIES) 1128
12.13 SCALING (M SERIES) 1138
12.14 COORDINATE SYSTEM ROTATION 1145
12.15 MACRO COMPILER/MACRO EXECUTER 1146
12.16 OPTIONAL ANGLE CHAMFERING AND CORNER ROUNDING (M SERIES) 1147
12.17 CHAMFERING AND CORNER ROUNDING (T SERIES) 1148
12.18 DIRECT DRAWING DIMENSIONS PROGRAMMING (T SERIES) 1150
12.19 PATTERN DATA INPUT 1152

13 DISPLAY/SET/EDIT 1166
13.1 DISPLAY/SET 1166
13.1.1 Run Hour and Parts Count Display 1166
13.1.2 Software Operator's Panel 1170
13.1.3 8-Level Data Protection Function 1178
13.1.4 Touch Panel Control 1183
13.1.5 External Touch Panel Interface 1188
13.1.6 Parameter Check Sum Function 1191
13.1.7 Touch Panel Check Signal 1201
13.1.8 CNC Screen Dual Display 1203
13.1.9 Speed Display Function of a Milling Tool with Servo Motor 1205
13.1.10 Screen Switching by Mode 1208
13.1.11 Screen Switching at Path Switching 1211
13.1.12 Screen erasure function and automatic screen erasure function 1212
13.1.13 Screen Hard Copy Function 1214
13.2 EDIT 1217
13.2.1 Memory Protection Keys 1217
13.2.2 Memory Protection Signal For CNC Parameter 1218
13.2.3 MDI Key Setting 1219
13.2.4 Compact-Type MDI Key Input Function 1220
13.3 TWO PATH DISPLAY AND EDIT 1222
13.3.1 Two Path Display 1222
13.3.2 Simultaneous Two Path Program Editing 1224
13.4 MACHINE OPERATION MENU 1228
13.4.1 Overview 1228
13.4.2 Explanation 1228
13.4.3 Parameter 1233
13.4.4 Signal 1234
13.4.5 Limitation 1234

13.5 MACHINE OPERATION MENU TOOL 1235
13.5.1 Overview 1235
13.5.2 Explanation 1235

14 INPUT/OUTPUT OF DATA 1243
14.1 READER/PUNCHER INTERFACE 1243
14.2 EXTERNAL I/O DEVICE CONTROL 1254

15 MEASUREMENT 1259
15.1 TOOL LENGTH MEASUREMENT (M SERIES) 1259
15.2 AUTOMATIC TOOL LENGTH MEASUREMENT (M SERIES) / AUTOMATIC TOOL OFFSET (T SERIES) 1260
15.3 SKIP FUNCTION 1267
15.3.1 Skip Function 1267
15.3.2 High-speed Skip Signal 1274
15.3.3 Multi-step Skip 1279
15.3.4 Torque Limit Skip Function 1286
15.4 COMPENSATION VALUE INPUT 1292
15.4.1 Input of tool offset value measured (T Series) 1292
15.4.2 Input of Tool Offset Value Measured B (T Series) 1293
15.4.3 Workpiece Origin Offset Measurement Value Direct Input 1306

16 PMC CONTROL FUNCTION 1309
16.1 PMC AXIS CONTROL 1309
16.1.1 PMC Axis Control 1309
16.2 EXTERNAL DATA INPUT 1377
16.3 EXTENDED EXTERNAL MACHINE ZERO POINT SHIFT 1390
16.4 EXTERNAL WORKPIECE NUMBER SEARCH 1393
16.5 EXTERNAL KEY INPUT 1395
16.6 ONE TOUCH MACRO CALL 1400

17 EMBEDDED ETHERNET FUNCTION 1407
17.1 EMBEDDED ETHERNET PORT AND PCMCIA ETHERNET CARD 1407
17.2 SETTING UP THE EMBEDDED ETHERNET FUNCTION 1408
17.2.1 Setting of the FOCAS2/Ethernet Function 1408
17.2.1.1 Operation on the FOCAS2/Ethernet setting screen 1408
17.2.1.2 Example of setting the FOCAS2/Ethernet function 1411
17.2.2 Setting of the FTP File Transfer Function 1411
17.2.2.1 Operation on the FTP file transfer setting screen 1412
17.2.2.2 Related NC parameters 1414
17.2.2.3 Example of setting the FTP file transfer function 1416
17.2.3 Setting Up the DNS/DHCP Function 1416
17.2.3.1 Setting up DNS 1416
17.2.3.2 Setting up DHCP 1417
17.2.3.3 Related NC parameters 1420
17.3 SWITCHING BETWEEN THE EMBEDDED ETHERNET DEVICES 1421
17.4 RESTART OF THE EMBEDDED ETHERNET 1421
17.5 MAINTENANCE SCREEN FOR EMBEDDED ETHERNET FUNCTION 1422
17.6 LOG SCREEN OF THE EMBEDDED ETHERNET FUNCTION 1426

18 DIAGNOSIS FUNCTION 1429
18.1 SERVO WARNING INTERFACE 1429

18.2 SPINDLE WARNING INTERFACE 1431
18.3 TROUBLE DIAGNOSIS 1433
18.3.1 Outline 1433
18.3.2 Trouble diagnosis guidance screen 1435
18.3.3 Trouble diagnosis monitor screen 1437
18.3.4 Trouble diagnosis parameter screen 1441
18.3.5 Trouble diagnosis graphic screen 1443
18.3.6 Trouble forecast level setting screen (only for servo axis) 1444
18.3.7 Parameter 1446
18.3.8 Signal 1447
18.3.9 Restrictions 1447
18.4 MACHINE ALARM DIAGNOSIS 1448
18.4.1 Outline 1448
18.4.2 Kind of additional alarm and operator message 1448
18.4.3 Available diagnosis number 1448
18.4.4 Environment for making trouble diagnosis message 1449
18.4.5 Guidance table for machine alarm diagnosis 1450
18.4.5.1 Install 1450
18.4.5.2 Uninstall 1450
18.4.5.3 Making a file to input trouble diagnosis messages 1451
18.4.5.4 Structure of the file to input trouble diagnosis messages 1452
18.4.6 Making trouble diagnosis messages 1453
18.4.6.1 Inputting guidance data 1454
18.4.6.2 Checking input data 1456
18.4.6.3 Making a memory card format file 1457
18.4.6.4 Jump from CNC guidance table to MTB's guidance table 1458
18.4.7 Making messages for multi-languages 1459
18.4.7.1 Making sheets for multi-languages 1459
18.4.7.2 Inputting data in the sheet for multi-languages 1461
18.4.8 Notice 1461
18.4.9 Translating data used with the former series(Series 0*i* /0*i* Mate-B/C, Series 16*i* /18*i* /21*i*-B) 1461

APPENDIX

A INTERFACE BETWEEN CNC AND PMC 1465
A.1 LIST OF ADDRESSES 1465
A.2 LIST OF SIGNALS 1502
A.2.1 List of Signals (In Order of Functions) 1502
A.2.2 List of Signals (In Order of Symbols) 1523
A.2.3 List of Signals (In Order of Addresses) 1541

B DIFFERENCES FROM Series 0*i*-C 1560
B.1 SETTING UNIT 1561
B.1.1 Differences in Specifications 1561
B.1.2 Differences in Signals 1561
B.1.3 Differences in Diagnosis Display 1561
B.2 STORED PITCH ERROR COMPENSATION 1562
B.2.1 Differences in Specifications 1562
B.2.2 Differences in Signals 1562
B.2.3 Differences in Diagnosis Display 1562
B.3 WORKPIECE COORDINATE SYSTEM 1563
B.3.1 Differences in Specifications 1563
B.3.2 Differences in Signals 1563

B.3.3 Differences in Diagnosis Display 1563
B.4 LOCAL COORDINATE SYSTEM 1564
B.4.1 Differences in Specifications 1564
B.4.2 Differences in Signals 1565
B.4.3 Differences in Diagnosis Display 1565
B.5 AXIS SYNCHRONOUS CONTROL 1565
B.5.1 Differences in Specifications 1565
B.5.2 Differences in Signals 1570
B.5.3 Differences in Diagnosis Display 1570
B.6 ARBITRARY ANGULAR AXIS CONTROL 1571
B.6.1 Differences in Specifications 1571
B.6.2 Differences in Signals 1571
B.6.3 Differences in Diagnosis Display 1571
B.7 STORED STROKE CHECK 1572
B.7.1 Differences in Specifications 1572
B.7.2 Differences in Signals 1573
B.7.3 Differences in Diagnosis Display 1573
B.8 CHUCK/TAIL STOCK BARRIER (T SERIES) 1574
B.8.1 Differences in Specifications 1574
B.8.2 Differences in Signals 1574
B.8.3 Differences in Diagnosis Display 1574
B.9 MACHINING CONDITION SELECTION FUNCTION 1574
B.9.1 Differences in Specifications 1574
B.9.2 Differences in Signals 1575
B.9.3 Differences in Diagnosis Display 1575
B.10 MANUAL HANDLE FEED 1576
B.10.1 Differences in Specifications 1576
B.10.2 Differences in Signals 1577
B.10.3 Differences in Diagnosis Display 1577
B.11 MANUAL REFERENCE POSITION RETURN 1577
B.11.1 Differences in Specifications 1577
B.11.2 Differences in Signals 1579
B.11.3 Differences in Diagnosis Display 1579
B.12 RESET AND REWIND 1580
B.12.1 Differences in Specifications 1580
B.12.2 Differences in Signals 1580
B.12.3 Differences in Diagnosis Display 1580
B.13 SINGLE DIRECTION POSITIONING (M SERIES) 1581
B.13.1 Differences in Specifications 1581
B.13.2 Differences in Signals 1581
B.13.3 Differences in Diagnosis Display 1581
B.14 MANUAL ABSOLUTE ON AND OFF 1581
B.14.1 Differences in Specifications 1581
B.14.2 Differences in Signals 1582
B.14.3 Differences in Diagnosis Display 1582
B.15 CIRCULAR INTERPOLATION 1583
B.15.1 Differences in Specifications 1583
B.15.2 Differences in Signals 1583
B.15.3 Differences in Diagnosis Display 1583
B.16 THREADING CYCLE RETRACT (CANNED CUTTING CYCLE/ MULTIPLE REPETITIVE CANNED CUTTING CYCLE) (T SERIES) 1583
B.16.1 Differences in Specifications 1583

B.16.2 Differences in Signals 1584
B.16.3 Differences in Diagnosis Display 1584
B.17 HELICAL INTERPOLATION 1584
B.17.1 Differences in Specifications 1584
B.17.2 Differences in Signals 1585
B.17.3 Differences in Diagnosis Display 1585
B.18 POLAR COORDINATE INTERPOLATION (T SERIES) 1585
B.18.1 Differences in Specifications 1585
B.18.2 Differences in Signals 1586
B.18.3 Differences in Diagnosis Display 1586
B.19 ADVANCED PREVIEW CONTROL (T SERIES)/AI ADVANCED PREVIEW CONTROL (M SERIES) / AI CONTOUR CONTROL (M SERIES) 1587
B.19.1 Differences in Specifications 1587
B.19.2 Differences in Signals 1589
B.19.3 Differences in Diagnosis Display 1589
B.20 WAITING M CODES (T SERIES (2-PATH CONTROL)) 1590
B.20.1 Differences in Specifications 1590
B.20.2 Differences in Signals 1590
B.20.3 Differences in Diagnosis Display 1590
B.21 PATH INTERFERENCE CHECK (T SERIES (2-PATH CONTROL)) 1591
B.21.1 Differences in Specifications 1591
B.21.2 Differences in Signals 1591
B.21.3 Differences in Diagnosis Display 1591
B.22 SYNCHRONOUS CONTROL AND COMPOSITE CONTROL (T SERIES (2-PATH CONTROL)) 1592
B.22.1 Differences in Specifications 1592
B.22.2 Differences in Signals 1596
B.22.3 Differences in Diagnosis Display 1596
B.23 SUPERIMPOSED CONTROL (T SERIES (2-PATH CONTROL)) 1596
B.23.1 Differences in Specifications 1596
B.23.2 Differences in Signals 1598
B.23.3 Differences in Diagnosis Display 1598
B.24 AUXILIARY FUNCTION/2ND AUXILIARY FUNCTION 1598
B.24.1 Differences in Specifications 1598
B.24.2 Differences in Signals 1598
B.24.3 Differences in Diagnosis Display 1598
B.25 SERIAL/ANALOG SPINDLE CONTROL 1599
B.25.1 Differences in Specifications 1599
B.25.2 Differences in Signals 1599
B.25.3 Differences in Diagnosis Display 1599
B.26 CONSTANT SURFACE SPEED CONTROL 1600
B.26.1 Differences in Specifications 1600
B.26.2 Differences in Signals 1600
B.26.3 Differences in Diagnosis Display 1600
B.27 SPINDLE POSITIONING (T SERIES) 1601
B.27.1 Differences in Specifications 1601
B.27.2 Differences in Signals 1601
B.27.3 Differences in Diagnosis Display 1602
B.28 Cs CONTOUR CONTROL 1602
B.28.1 Differences in Specifications 1602
B.28.2 Differences in Signals 1602

B.28.3 Differences in Diagnosis Display 1602
B.29 MULTI-SPINDLE CONTROL 1603
B.29.1 Differences in Specifications 1603
B.29.2 Differences in Signals 1603
B.29.3 Differences in Diagnosis Display 1603
B.30 TOOL FUNCTIONS 1604
B.30.1 Differences in Specifications 1604
B.30.2 Differences in Signals 1605
B.30.3 Differences in Diagnosis Display 1605
B.31 TOOL COMPENSATION MEMORY 1606
B.31.1 Differences in Specifications 1606
B.31.2 Differences in Signals 1607
B.31.3 Differences in Diagnosis Display 1607
B.32 Y AXIS OFFSET (T SERIES) 1607
B.32.1 Differences in Specifications 1607
B.32.2 Differences in Signals 1607
B.32.3 Differences in Diagnosis Display 1607
B.33 CUTTER COMPENSATION/TOOL NOSE RADIUS COMPENSATION .. 1608
B.33.1 Differences in Specifications 1608
B.33.2 Differences in Signals 1613
B.33.3 Differences in Diagnosis Display 1613
B.34 CUSTOM MACRO 1614
B.34.1 Differences in Specifications 1614
B.34.2 Differences in Signals 1616
B.34.3 Differences in Diagnosis Display 1616
B.34.4 Miscellaneous 1616
B.35 INTERRUPTION TYPE CUSTOM MACRO 1616
B.35.1 Differences in Specifications 1616
B.35.2 Differences in Signals 1616
B.35.3 Differences in Diagnosis Display 1616
B.36 CANNED CYCLE FOR DRILLING 1617
B.36.1 Differences in Specifications 1617
B.36.2 Differences in Signals 1619
B.36.3 Differences in Diagnosis Display 1619
B.37 CANNED CYCLE (T SERIES)/MULTIPLE REPETITIVE CANNED CYCLE (T SERIES) 1619
B.37.1 Differences in Specifications 1619
B.37.2 Differences in Signals 1620
B.37.3 Differences in Diagnosis Display 1620
B.38 CANNED GRINDING CYCLE 1620
B.38.1 Differences in Specifications 1620
B.38.2 Differences in Signals 1621
B.38.3 Differences in Diagnosis Display 1621
B.39 MULTIPLE RESPECTIVE CANNED CYCLE FOR TURNING (T SERIES) 1621
B.39.1 Differences in Specifications 1621
B.39.2 Differences in Signals 1625
B.39.3 Differences in Diagnosis Display 1625
B.40 OPTIONAL ANGLE CHAMFERING AND CORNER ROUNDING (M SERIES) 1625
B.40.1 Differences in Specifications 1625
B.40.2 Differences in Signals 1626

B.40.3 Differences in Diagnosis Display 1626
B.41 CHAMFERING AND CORNER ROUNDING (T SERIES) 1626
B.41.1 Differences in Specifications 1626
B.41.2 Differences in Signals 1626
B.41.3 Differences in Diagnosis Display 1626
B.42 DIRECT DRAWING DIMENSIONS PROGRAMMING (T SERIES) 1627
B.42.1 Differences in Specifications 1627
B.42.2 Differences in Signals 1627
B.42.3 Differences in Diagnosis Display 1627
B.43 RUN HOUR AND PARTS COUNT DISPLAY 1627
B.43.1 Differences in Specifications 1627
B.43.2 Differences in Signals 1628
B.43.3 Differences in Diagnosis Display 1628
B.44 SCREEN ERASURE FUNCTION AND AUTOMATIC SCREEN ERASURE FUNCTION 1628
B.44.1 Differences in Specifications 1628
B.44.2 Differences in Signals 1629
B.44.3 Differences in Diagnosis Display 1629
B.45 MEMORY PROTECTION SIGNAL FOR CNC PARAMETER 1629
B.45.1 Differences in Specifications 1629
B.45.2 Differences in Signals 1629
B.45.3 Differences in Diagnosis Display 1629
B.46 AUTOMATIC TOOL LENGTH MEASUREMENT (M SERIES)/ AUTOMATIC TOOL OFFSET (T SERIES) 1630
B.46.1 Automatic Tool Length Measurement (M Series) 1630
B.46.1.1 Differences in Specifications 1630
B.46.1.2 Differences in Signals 1631
B.46.1.3 Differences in Diagnosis Display 1631
B.46.2 Automatic Tool Offset (T Series) 1631
B.46.2.1 Differences in Specifications 1631
B.46.2.2 Differences in Signals 1632
B.46.2.3 Differences in Diagnosis Display 1632
B.47 SKIP FUNCTION 1633
B.47.1 Differences in Specifications 1633
B.47.2 Differences in Signals 1634
B.47.3 Differences in Diagnosis Display 1635
B.48 INPUT OF TOOL OFFSET VALUE MEASURED B (T SERIES) 1635
B.48.1 Differences in Specifications 1635
B.48.2 Differences in Signals 1635
B.48.3 Differences in Diagnosis Display 1636
B.49 PMC AXIS CONTROL 1636
B.49.1 Differences in Specifications 1636
B.49.2 Differences in Signals 1641
B.49.3 Differences in Diagnosis Display 1641
B.50 EXTERNAL DATA INPUT 1642
B.50.1 Differences in Specifications 1642
B.50.2 Differences in Signals 1644
B.50.3 Differences in Diagnosis Display 1644
B.51 SEQUENCE NUMBER SEARCH 1645
B.51.1 Differences in Specifications 1645
B.51.2 Differences in Signals 1645
B.51.3 Differences in Diagnosis Display 1645
B.52 IN-POSITION CHECK 1645

B.52.1 Differences in Specifications....1645
B.52.2 Differences in Signals....1646
B.52.3 Differences in Diagnosis Display....1646
B.53 DATA SERVER FUNCTION....1646
B.53.1 Differences in Specifications....1646
B.53.2 Differences in Signals....1646
B.53.3 Differences in Diagnosis Display....1646
B.54 POWER MATE CNC MANAGER....1647
B.54.1 Differences in Specifications....1647
B.54.2 Differences in Signals....1647
B.54.3 Differences in Diagnosis Display....1647
B.55 PROGRAMMABLE PARAMETER INPUT (G10)....1647
B.55.1 Differences in Specifications....1647
B.55.2 Differences in Signals....1647
B.55.3 Differences in Diagnosis Display....1647
B.56 EXTERNAL SUBPROGRAM CALL (M198)....1648
B.56.1 Differences in Specifications....1648
B.56.2 Differences in Signals....1648
B.56.3 Differences in Diagnosis Display....1648

1 AXIS CONTROL

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

1.1 CONTROLLED AXIS 1
1.2 SETTING EACH AXIS 2
1.3 ERROR COMPENSATION 20
1.4 SETTINGS RELATED TO SERVO-CONTROLLED AXES 48
1.5 SETTINGS RELATED WITH COORDINATE SYSTEMS 92
1.6 AXIS SYNCHRONOUS CONTROL 113
1.7 TANDEM CONTROL 136
1.8 ARBITRARY ANGULAR AXIS CONTROL 144
1.9 ELECTRONIC GEAR BOX (M SERIES) 156

1.1 CONTROLLED AXIS

Overview

The number of controlled paths and the total number of controlled axes depend on the model as shown below.

Series 0*i* -D

Item	M series	T series	
Controlled path	1 path	1 path	2 paths
Total number of controlled axes (Feed axes + Spindle axes)	Max. 8 axes	Max. 8 axes	Max. 11 axes (Total of two path)
Total feed axes	Max. 7 axes	Max. 7 axes	Max. 9 axes (Total of two path)
Feed axes (for each path)	Max. 7 axes	Max. 7 axes	Max. 7 axes
Simultaneously controlled axes (for each path)	Max. 4 axes	Max. 4 axes	Max. 4 axes
Axis control by PMC	Max. 4 axes at a time (Not available on Cs axis)	Max. 4 axes at a time (Not available on Cs axis)	Max. 4 axes at a time (Not available on Cs axis)
Designation of spindle axes (each path/total)	2 axes	Max. 3 axes	Max. 3 axes/4 axes
Cs contour control (each path/total)	2 axes	Max. 3 axes	Max. 3 axes/4 axes

Series 0*i* Mate -D

Item	M series	T series
Controlled path	1 path	1 path
Total number of controlled axes (Feed axes + Spindle axes)	Max. 5 axes	Max. 5 axes
Total feed axes	Max. 4 axes	Max. 3 axes
Feed axes (for each path)	Max. 4 axes	Max. 3 axes
Simultaneously controlled axes	Max. 3 axes	Max. 3 axes
Axis control by PMC	Max. 4 axes at a time	Max. 3 axes at a time (Not available on Cs axis)
Designation of spindle axes	1 axis	Max. 2 axes
Cs contour control	-	Max. 1 axis

NOTE
The maximum numbers above vary with the option configuration. For details, refer to the manual provided by the machine tool builder.

Alarm and message

Number	Message	Description
PS0015	TOO MANY SIMULTANEOUS AXES	A move command was specified for more axes than can be controlled by simultaneous axis control. Divide the program-specified moving axis into two blocks.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Number of controlled axes
CONNECTION MANUAL (FUNCTION) (This manual)	2-path control
CONNECTION MANUAL (FUNCTION) (This manual)	Designation of spindle axes

1.2 SETTING EACH AXIS

1.2.1 Name of Axes

Overview

Each axis that is controlled by the CNC (including those controlled by the PMC) must be named. Select and set names from among X, Y, Z, A, B, C, U, V, and W (with parameter 1020).
The names of the basic axes, however, are fixed (X, Y, and Z for the M series and X and Z for the T series). The names of additional axes can be selected, as desired, from the names other than those for the basic axes.
The same name cannot be assigned to more than one axis.
For 2-path control, the names of the basic axes in each path are fixed. The names of additional axes can be selected arbitrarily, using parameter No. 1020, from the names other than those of the basic axes. While one path is not allowed to have two or more axes with the same name, more than one axis can have the same name if each belongs to a different path.

Parameter

1020	Program axis name for each axis

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 67,85 to 90
An axis name (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 T series, however, 'U', 'V', and 'W' are not selectable.)

(Tip) ASCII code

Axis name	X	Y	Z	A	B	C	U	V	W
Setting	88	89	90	65	66	67	85	86	87

For the axes with axis names of 'X', 'Y', 'Z', and 'C' in G code system A of the T series, the 'U', 'V', 'W', and 'H' commands are the incremental commands of these axes.

NOTE
1 When G code system A is used in the T series, U, V, or W cannot be used as an axis name.
2 The same axis name cannot be set for multiple axes.
4 When address C or A is used during chamfering/corner rounding or direct drawing dimension programming (when bit 4 (CCR) of parameter No. 3405 is 1) in the T series, address C or A cannot be used as an axis name.
5 When the multiple repetitive turning canned cycle (T series) is used, only 'X', 'Y', and 'Z' can be used for the address of the target axis.

3131	Subscript of axis name

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 9, 32, 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 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

When the subscription of an axis name is not set in a 2-path system, the subscription of an axis name is automatically set to the path number. To hide the subscription of an axis name, set the parameter of the subscription of an axis name to the ASCII code (32) of a space.

Alarm and message

Number	Message	Description
PS0009	IMPROPER NC-ADDRESS	An illegal address was specified, or parameter 1020 is not set.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Axis name

1.2.2 Increment System

Overview

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 degrees.

There are three types of increment systems as listed in Table 1.2.2(a). A desired type can be set for each axis using the corresponding parameter (ISA or ISC (bit 0 or 1 of parameter No. 1013)).

The least input increment can be set to metric input or inch input using the G code (G20 or G21) or setting parameter (bit 2 (INI) of parameter No. 0000).

The least command increment is set to either metric or inch system depending on the machine tool in advance. Select the metric or inch system using bit 0 (INM) of parameter No. 1001 in advance.

Any combined use of the inch and metric systems is not allowed. There are functions that cannot be used across axes with different increment systems (such as circular interpolation and cutter compensation).

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-64302EN).

Table 1.2.2 (a) Increment system

Name of an increment system	Least input increment		Least command increment		Maximum stroke	
IS-A	0.01	mm	0.01	mm	±999999.99	mm
	0.001	inch	0.001	inch	±99999.999	inch
	0.01	deg	0.01	deg	±999999.99	deg
IS-B	0.001	mm	0.001	mm	±999999.999	mm
	0.0001	inch	0.0001	inch	±99999.9999	inch
	0.001	deg	0.001	deg	±999999.999	deg
IS-C	0.0001	mm	0.0001	mm	±99999.9999	mm
	0.00001	inch	0.00001	inch	±9999.99999	inch
	0.0001	deg	0.0001	deg	±99999.9999	deg

When bit 7 (IPR) of parameter No. 1004, which multiplies the input increment by 10, is set to 1 and a value is specified with no decimal point, the specifications of each increment system are changed as listed in Table 1.2.2(b).

Table 1.2.2 (b)

Name of an increment system	Least input increment		Least command increment		Maximum stroke	
IS-B	0.01	mm	0.001	mm	±999999.999	mm
	0.001	inch	0.0001	inch	±99999.9999	inch
	0.01	deg	0.001	deg	±999999.999	deg
IS-C	0.001	mm	0.0001	mm	±99999.9999	mm
	0.0001	inch	0.00001	inch	±9999.99999	inch
	0.001	deg	0.0001	deg	±99999.9999	deg

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0000						INI		

[Input type] Setting input
[Data type] Bit path

 # 2 INI Unit of input
0: In metrics
1: In inches

	#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
1004	IPR							

[Input type] Parameter input
[Data type] Bit path

#7 IPR Whether the least input increment for each axis is set to a value 10 times as large as the least command increment is specified, in increment systems of IS-B or IS-C at setting mm.
0: The least input increment is not set to a value 10 times as large as the least command increment.
1: The least input increment is set to a value 10 times as large as the least command increment.
If IPR is set to 1, the least input increment is set as follows:

Input increment	Least input increment
IS-B	0.01 mm, 0.01 deg, or 0.0001 inch
IS-C	0.001 mm, 0.001 deg, or 0.00001 inch

NOTE
For IS-A, the least input increment cannot be set to a value 10 times as large as the least command increment.
The least input increment is not multiplied by 10 also when the calculator-type decimal point input (bit 0 (DPI) of parameter No. 3401) is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1006					DIAx			

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#3 DIAx The move command for each axis is based on:
0: Radius specification
1: Diameter specification

NOTE
For the FS0*i*-C, one of the following changes is required besides setting bit 3 (DIAx) of parameter No. 1006 so that the axis based on diameter specification achieves the specified amount of movement.
- Halve the command multiplication (the detection unit is not changed).
- Halve the detection unit and double the flexible feed gear (DMR).

For the FS0*i*-D, only if bit 3 (DIAx) of parameter No. 1006 is set, the CNC halves the specified pulse. Accordingly, the above changes are not required (when the detection unit is not changed). To halve the detection unit, double both CMR and DMR.

	#7	#6	#5	#4	#3	#2	#1	#0
1013							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 ISA
#1 ISC Increment system of each axis

Increment system	#1 ISCx	#0 ISAx
IS-A	0	1
IS-B	0	0
IS-C	1	0

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Increment system

1.2.3 Specifying the Rotation Axis

Overview

Bit 0 (ROTx) of parameter 1006 can be used to set each axis to a linear axis or rotation axis. Bit 1 (ROSx) of parameter 1006 can be used to select the rotation axis type, A or B, for each axis. See the explanation of the parameters for details of types A and B.

When the roll over function is used, the values displayed for absolute coordinates are rounded by the shift amount per rotation, as set in parameter No. 1260. This can prevent coordinates for the rotation axis from overflowing. 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. The roll–over function is enabled by setting bit 0 (ROAx) of parameter 1008 to 1.

For an absolute command, the coordinates after the tool has moved are values rounded by the angle corresponding to one rotation set in parameter No. 1260. 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. For an incremental command, the tool moves the angle specified in the command.

NOTE
Rotary axis roll–over function cannot be used together with the indexing function (M series) of the index table. To disable the indexing function (M series), set 1 in bit 0 (ITI) of parameter No. 5501 or 0 in bit 3 (IXC) of parameter No. 8132.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1006							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 rotation axis.
#1 ROSx

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	Rotation axis (A type) (1)Inch/metric conversion is not done. Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by parameter No.1008#0(ROAx) and #2(RRLx). (2)Stored pitch error compensation is the rotation type. (Refer to parameter No.3624) (3)Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.

ROSx	ROTx	Meaning
1	1	Rotation axis (B type) (1)Inch/metric conversion, absolute coordinate values and relative coordinate values are 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 rotation axis roll over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

	#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 roll-over function of a rotation axis is
0: Invalid
1: Valid

NOTE
ROAx specifies the function only for a rotation axis (for which ROTx, #0 of parameter No.1006, is set to 1)

#1 RABx In the absolute commands, 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.

1260	Amount of a shift per one rotation of a rotation 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 amount of a shift per one rotation of a rotation axis.
For the rotation axis used for cylindrical interpolation, set the standard value.

	#7	#6	#5	#4	#3	#2	#1	#0
5501								
								ITI

[Input type] Parameter input
[Data type] Bit path

#0 ITI The index table indexing function (M series) is:
0: Enabled.
1: Disabled.

NOTE
To enable the index table indexing function, set bit 3 (IXC) of parameter No. 8132 to 1 in addition to this parameter. The index table indexing function is enabled only when both ITI and IXC are enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
8132								
					IXC			

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#3 IXC Index table indexing function (M series) is:
0: Not Used.
1: Used.

NOTE
When enabling the index table indexing function, set bit 0 (ITI) of parameter No. 5501 to 0 in addition to this parameter. The index table indexing function is enabled only when both ITI and IXC are enabled.

Note

NOTE
Rotary axis roll–over function cannot be used together with the indexing function (M series) of the index table.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Rotary axis roll-over function

1.2.4 Controlled Axes Detach

Overview

These signals release the specified control axes from control by the CNC.
When attachments are used (such as a detachable rotary table), these signals are selected according to whether the attachments are mounted.
The signals can also be used for switching the C axis and spindle on lathes.
When multiple rotary tables are used in turn, the tables must use motors of the same model. Absolute pulse coders cannot be used.

> ⚠ **WARNING**
> For a vertical axis, in particular, it is necessary to prepare a sequence that starts operating the mechanical brake before the control axis detach operation.
> When this method is applied to a vertical axis, special care should be taken.

Signal

Controlled axis detach signals DTCH1 to DTCH5<Gn124.0 to Gn124.4>

[Classification] Input signal

[Function] These signals detach the control axes from control.
These signals are provided for each control axis; the affixed number of the signal name shows the control axis number.
DTCHx
x : 1 ..... The 1st axis is detached.
2 ..... The 2nd axis is detached.
3 ..... The 3rd axis is detached.
: :

[Operation] When the signals are 1, the control unit operates as follows:
<1> Position control is not executed at all. Servo motor excitation is cut.
<2> Servo alarm on the axis is ignored.
<3> Axis interlock signal is assumed to be zero on the detached axis.
<4> A command for automatic or manual operation for the axis does not cause an alarm, but the operation is restrained because the axis interlock signal is 0. In an automatic operation, the execution may stop and hold at the block. Do not execute any command for automatic or manual operation for the axis.
<5> Position display also displays the position of the detached axis.

Controlled axis detach status signals MDTCH1 to MDTCH5<Fn110.0 to Fn110.4>

[Classification] Output signal

[Function] These signals notify the PMC that the corresponding axes have been released from control.
These signals are provided for each control axis; the affixed number of the signal name shows the control axis number.

MDTCHx
x : 1 ..... The 1st axis is detached.
2 ..... The 2nd axis is detached.
3 ..... The 3rd axis is detached.
: :

[Output cond.] These signals are 1 in the following case:
- When the corresponding axes are released from control

These signals are 0 in the following case:
- When the corresponding axes are under control

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn124				DTCH5	DTCH4	DTCH3	DTCH2	DTCH1
Fn110				MDTCH5	MDTCH4	MDTCH3	MDTCH2	MDTCH1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0012	RMVx							

[Input type] Setting input
[Data type] Bit axis

#7 RMVx Releasing the assignment of the control axis for each axis
0: Not released
1: Released
(Equivalent to the control axis detachment signals DTCH1, DTCH2, and so forth)

> **NOTE**
> RMVx is valid when bit 7 (RMBx) of parameter No. 1005 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1005	RMBx	MCCx						

[Input type] Parameter input
[Data type] Bit axis

#6 MCCx If a multi-axis amplifier is used, and another axis of the same amplifier is placed in the control axis detach state, the MCC signal of the servo amplifier is:
0: Turned off.
1: Not turned off. (The servo motor is deactivated, but the MCC signal of the servo amplifier is not turned off.)

> **⚠ WARNING**
> 1 When this parameter is set to 1, the dynamic brake does not operate during removal of an axis. Therefore, if a failure occurs in the mechanical brake, driving circuit, or sequence, a vertical axis may fall freely in a significant manner. Since an excess error check cannot also be performed during removal of an axis, set this parameter to 0 for a vertical axis.

⚠ WARNING

2 When the servo motor of a controlled axis to be detached is connected to a multi-axis amplifier such as a two-axis amplifier, placing the axis in the control axis detach state causes the activating current in the amplifier to drop. As a result, alarm (SV0401) "V READY OFF" is issued in the other axes. This alarm can be suppressed by setting this parameter bit.
With this method, however, the target axis for the control axis detach operation is placed in the servo off state (the amplifier remains on, but no current flows through the motor). The torque of the target axis becomes 0, so care should be taken. Even when a controlled axis has been detached, detaching a cable (a command cable or feedback cable) of the axis causes an alarm.
In such applications, it is impossible to perform a control axis detach operation with a multi-axis amplifier by setting this parameter bit. (Prepare a single-axis amplifier.)

NOTE

This parameter can be set for an axis to be controlled.
This parameter is used to remove only one axis from a two-axis amplifier or three-axis amplifier.
When only one axis is removed from a two-axis amplifier or three-axis amplifier, servo alarm SV0401 "V-READY OFF" is issued. Setting parameter to 1 prevents servo alarm SV0401 "V-READY OFF" from being issued.
When the CNC is disconnected from the servo, however, "V-READY OFF" is issued by the servo amplifier because of the characteristic of a multi-axis amplifier.

#7 **RMBx** The control axis detachment signal for each axis and the setting input RMV (bit 7 of parameter No. 0012) are:

0: Invalid
1: Valid

Caution

⚠ CAUTION

When a multiaxis amplifier is used, the motor cannot be disconnected from the amplifier. When the motor needs to be disconnected from the amplifier for replacement of the rotary table or other reasons, a 1-axis amplifier must be used.

Note

NOTE

1 Controlled axis detach signals DTCH1 <G124.0>, DTCH2 <G124.1>, DTCH3 <G124.2>, 0 can be changed from 1 to 0 or from 0 to 1 when the power is first turned on or when no movement is being executed along the corresponding axis. If these signals are changed from 0 to 1 when the tool is moving along the corresponding axis, the axis is released from control upon completion of the movement.

NOTE
2 For these signals to be attached, parameter RMB (No. 1005#7) must be set, indicating the axes are detachable.
3 Setting parameter RMV (No. 0012#7) from the MDI panel detaches the axes in the same way as these signals.
4 Those axes that are released from control lose their reference positions. Reference position return must, therefore, be performed for the axes prior to executing move commands for the axes. Specifying a move command before reference position return has been performed causes alarm PS0224 to be output.
(Setting bit 0 (ZRN) of parameter No. 1005 can prevent alarm PS0224 from occurring.)
If an axis for which an absolute position detector is used (bit 5 (APC) of parameter No. 1815 is set to 1) is released from control, the correspondence between the machine position and reference position is lost. Consequently, bit 4 (APZ) of parameter No. 1815 indicating that the correspondence is established is set to 0, resulting in alarm DS0300. After an axis is released from control, perform reference position return to bring the machine position into correspondence with the reference position.

1.2.5 Outputting the Movement State of an Axis

Overview

The movement state of each axis can be output to the PMC.

Signal

Axis moving signals MV1 to V5<Fn102.0 to Fn102.4>

[Classification] Output signal

[Function] These signals indicate that a control axis is moving.
The signals are provided for each control axis, and the number in the signal name corresponds to the control axis number.
MVx
x : 1 ..... The 1st axis is moving.
2 ..... The 2nd axis is moving.
3 ..... The 3rd axis is moving.
: :
: :

[Output cond.] The signals turn to "1" in the following cases:
- The corresponding axis has started moving.
- In manual handle feed mode, the handle feed axis of the corresponding axis has been selected.

The signals turn to "0" in the following case:
- The corresponding axis has stopped moving and enters the in-position status.

Axis moving direction signals MVD1 to MVD5<Fn106.0 to Fn106.4>

[Classification] Output signal

[Function] These signals indicate the movement direction of control axis.
They are provided for each control axis, and the number in the signal name corresponds to the control axis number.

MVDx
x : 1 ..... The moving direction of the 1st axis is minus.
2 ..... The moving direction of the 2nd axis is minus.
3 ..... The moving direction of the 3rd axis is minus.
: :
: :

[Output cond.] "1" indicates the corresponding axes are moving in the minus direction, and "0" indicates they are moving in the plus direction.

> **⚠ CAUTION**
> These signals maintain their condition during a stop, indicating the direction of the axes' movement before stopping.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn102				MV5	MV4	MV3	MV2	MV1
Fn106				MVD5	MVD4	MVD3	MVD2	MVD1

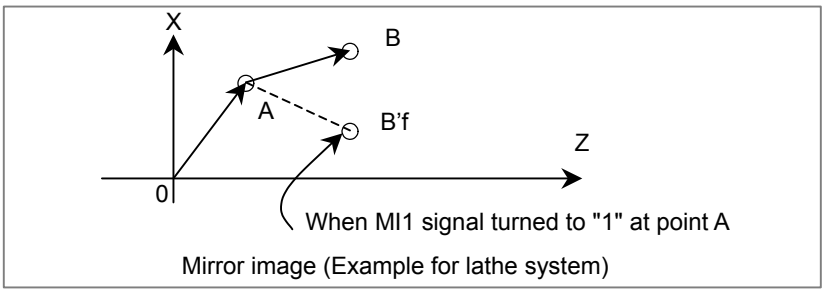
CAUTION

> **⚠ CAUTION**
> Axis moving signals and axis moving direction signals are output in both automatic and manual operations.

1.2.6 Mirror Image

Overview

Mirror image can be applied to each axis, either by signals or by parameters (setting input is acceptable). All movement directions are reversed during automatic operation along axes to which a mirror image is applied.



Mirror image (Example for lathe system)

However, the following directions are not reversed:

- Direction of manual operation and direction of movement, from the intermediate position to the reference position during automatic reference position return (for the M series and T series)
- Approach direction for single direction positioning (G60) and shift direction for boring cycles (G76 and G87) (for M series only)

Mirror image check signals indicate whether mirror image is applied to each axis. System variable #3007 contains the same information (refer to the Operator's Manual).

Signal

Mirror image signals MI1 to MI5<Gn106.0 to Gn106.4>

[Classification] Input signal
[Function] Apply mirror image to the specified axes.
[Operation] Apply mirror image to those axes for which the signals are 1.

These signals are provided for the controlled axes on a one-to-one basis. A number appended to a signal represents the controlled axis number.

MIx

x : 1 ..... Applies mirror image to the 1st axis.
2 ..... Applies mirror image to the 2nd axis.
3 ..... Applies mirror image to the 3rd axis.
: :
: :

The mirror image signal can be turned to "1" in the following cases:

(1) During offset cancel;
(2) When the CNC is in the automatic operation stop state and not in the feed hold state.

Mirror image check signals MMI1 to MMI5<Fn108.0 to Fn108.4>

[Classification] Output signal

[Function] These signals indicate the mirror image condition of each axis.

The mirror image is set by taking the logical sum of the signal from the MDI panel and the input signal of the machine tool, then relaying the information to the machine tool.

These signals are provided for every control axis; the numeral in the signal name indicates the relevant control axis number.

MMIx

x : 1 ..... Mirror image is applied to the 1st axis
2 ..... Mirror image is applied to the 2nd axis
3 ..... Mirror image is applied to the 3rd axis
: :
: :

[Output cond.] These signals turn to "1" when:

- Mirror image signal MIn of the corresponding axis is "1"; or
- Mirror image of the corresponding axis is turned on by setting data from the MDI panel.

These signals turn to "0" when:

- Mirror image signal (MIn) of the corresponding axis is "0" and the setting of the mirror image in the control unit is turned off.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn106				MI5	MI4	MI3	MI2	MI1
Fn108				MMI5	MMI4	MMI3	MMI2	MMI1

Parameter

	#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)

Warning

⚠ WARNING

1 When programmable mirror image (M series) and ordinary mirror image are specified at the same time, programmable mirror image is applied first.
2 No programmable mirror image (M series) affects mirror image check signals MMI1 to MMI5 <F108>.

CAUTION

⚠ CAUTION

Even when the mirror image is applied, commands which do not actuate mirror image (such as automatic reference position return and manual operation) do not affect mirror image check signals MMI1 to MMI5 <F108>.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Mirror image

1.2.7 Follow-up

Overview

If the machine moves in the state in which position control on controlled axes is disabled (during servo-off, emergency stop, or servo alarm), feedback pulses are accumulated in the error counter. The CNC reflects the machine movement corresponding to the error count in the current position managed by the CNC. This operation is referred to as follow-up. When follow-up is performed, the current position managed by the CNC does not shift from the actual machine position.
You can select whether to perform follow-up for axes when the servo is turned off.
Follow-up is always performed during emergency stop or a servo alarm.

Explanation

- When follow-up is not performed for the axes for which the servo is turned off

When follow-up signal *FLWU is 1 or bit 0 (FUPx) of parameter 1819 is 1, follow-up is not performed. The error is added to the error counter as a servo error.
In this case, the machine moves to compensate for the error when the servo off signal changes to 0.
In general, follow-up is not used if the machine is mechanically clamped when position control is disabled for the controlled axes.

- When follow-up is performed for the axes for which the servo is turned off

When follow-up signal *FLWU is "0", the follow-up function is engaged. The present position of the CNC is changed to reset the error counter to zero. The machine tool remains in a deviated position, but since the present position of the CNC changes correspondingly, the machine moves to the correct position when the absolute command is next applied.
In general, follow-up should be used when motors are driven by mechanical handles.

⚠ CAUTION

If follow-up is performed after the servo is turned off during automatic operation, the limitation applies that is described in item 3 in CAUTION of the next section "Servo Off/Mechanical Handle Feed".

Signal

Follow-up signal *FLWU<Gn007.5>

[Classification] Input signal

[Function] Select whether to perform follow-up when the servo is turned off for those axes for which bit 0 (FUPx) of parameter 1819 is 0.

[Operation] 0: Performs follow-up.
1: Does not perform follow-up.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn007			*FLWU					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1819								FUPx

[Input type] Parameter input

[Data type] Bit axis

#0 FUPx To perform follow-up when the servo is off is set for each axis.

0: The follow-up signal, *FLWU, determines whether follow-up is performed or not.
When *FLWU is 0, follow-up is performed.
When *FLWU is 1, follow-up is not performed.

1: Follow-up is not performed.

NOTE
When using the index table indexing function (M series), set FUPx to 1 for a control axis subject to index table indexing.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (this manual)	Servo off/Mechanical Handle Feed

1.2.8 Servo off/Mechanical Handle Feed

Overview

Place the controlled axes in the servo off state, stop the current to the servo motor, which disables position control. However, the position detection feature functions continuously, so the current position is not lost.

These signals are used to prevent the servo motors from overloading when the tools on the axes are mechanically clamped under certain machining conditions on the machine, or to move the machine by driving the motors by mechanical handles.

Signal

Servo off signals SVF1 to SVF5<Gn126.0~Gn126.4>

[Classification] Input signal

[Function] Select whether to place each axis in the servo off state.
These signals are provided for the controlled axes on a single axis basis. A number appended to a signal represents a controlled axis number.

SVFx
x : 1 ..... Servo off for the first axis
2 ..... Servo off for the second axis
3 ..... Servo off for the third axis
: :

[Operation] These signals put the axes for which the signals are 1 in the servo off state (the current to the servo motor is stopped). This disables position control. However, the position detection feature continues to function, so the current position is not lost.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn126				SVF5	SVF4	SVF3	SVF2	SVF1

Warning

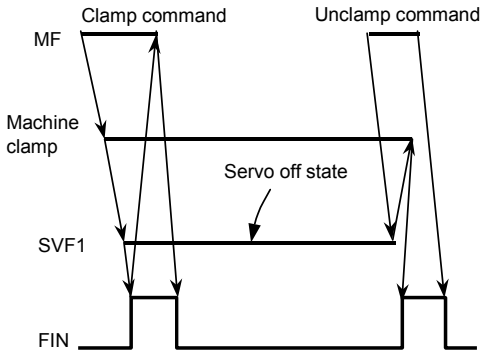
⚠ WARNING

In the servo off state, the motor is deactivated and the dynamic brake does not work. This leaves the servo motor completely free with no braking power.
The vertical axis drops when placed in the servo off state. Apply the mechanical break before inputting the servo off signal. Select not to perform follow-up when the servo is turned off and make an excess error check during the servo off state, in order to prevent the axis from dropping freely in the event of a failure of the mechanical break or its drive circuit, sequence, etc.
The horizontal axis also needs to be stopped before it can be placed in the servo off state.

Caution

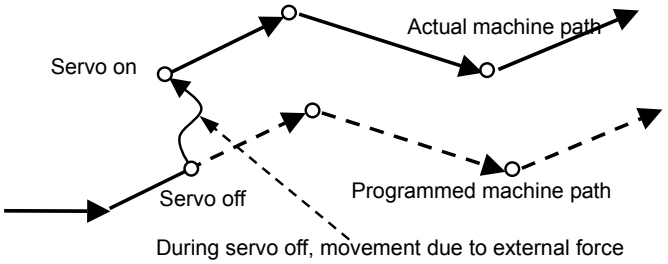
⚠ CAUTION

1 In general, interlock is applied to an axis while the servo off signal for that axis is 1.
2 When one of these signals turns to "1", the servo motor is turned off.
The mechanical clamp is done by using the auxiliary function. Set the timing for the auxiliary function, mechanical clamp and servo off signals as shown in the diagram below. The clamp command auxiliary function should be executed only after the distribution end signal (DEN) turned to "1".



⚠ CAUTION

3 If, during automatic operation, a servo off signal is issued with the setting that causes follow-up to be performed (*FLWU<Gn007.5> = ’0’), even if the machine is moved with external force or other means, the travel distance will not immediately reflected in coordinates. Until it is reflected, the coordinates will shift by the amount of movement due to the external force, and the subsequent machine path will be as in the figure below.



The following method is available to reflect the amount of movement during servo off in coordinates. If not wishing to shift the path, be sure to follow the procedure below to adjust the coordinates and execute an absolute command.

- Exit from auto automatic operation with a reset, single block stop, or feed hold, and then make a restart.

4 If a servo off signal is issued with the setting that does not cause follow-up to be performed (*FLWU<Gn007.5> = ’1’), even if the machine is moved with external force or other means, the machine will be retracted by the travel distance in the servo on state and, therefore, the path will never shift in subsequent automatic operation.
The amount of movement due to the external force in the servo off state is regarded as a servo positional deviation and is stored inside the NC. Thus, when a servo on signal is issued, axis moving occurs to cancel this servo positional deviation. The machine moves at a speed in accordance with the servo loop gain and if the amount is large, this may give the machine a shock.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (this manual)	Follow-up

1.2.9 Position Switch

Overview

Position switch signals can be output to the PMC while the machine coordinates along a controlled axes are within a specified ranges.
Using parameters, specify arbitrary controlled axes and machine coordinate operating ranges for which position switch signals are output.
Up to 16 position switch signals can be output.
(When 11 or more position switch signals are used, set 1 in bit 1 (EPW) of parameter No. 6901.)

⚠ CAUTION

The position switch function is enabled after reference position return is completed.

Signal

Position switch signals PSW01 to PSW16<Fn070, Fn071>

[Classification] Output signal

[Function] Indicates that the machine coordinates along the controlled axes specified by parameters Nos. 6910 to 6925 are within the ranges specified by parameters Nos. 6930 to 6945 and 6950 to 6965.
The position switch signal corresponding to the n-th position switch function is PSWn.
(n : 1 to 16)

[Output cond.] These signals are 1 in the following case:
- When the machine coordinates along the controlled axes are within the specified ranges.

These signals are 0 in the following case:
- When the machine coordinates along the controlled axes are not within the specified ranges.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn070	PSW08	PSW07	PSW06	PSW05	PSW04	PSW03	PSW02	PSW01
Fn071	PSW16	PSW15	PSW14	PSW13	PSW12	PSW11	PSW10	PSW09

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6901						PSA	EPW	

[Input type] Parameter input
[Data type] Bit path

#1 EPW The maximum number of points of the position switch is:
0: 10.
1: 16.

#2 PSA In determination of a position switch function operation range, a servo delay amount (positional deviation) and a delay amount in acceleration/deceleration control are:
0: Not considered.
1: Considered.

6910	Controlled axis for which the 1-st position switch function is performed
to	
6925	Controlled axis for which the 16-th position switch function is performed

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes
Set the controlled axis number corresponding to one of the first to sixteenth position switch functions. When the machine coordinate of the corresponding axis is within a parameter-set range, the corresponding position switch signal is output to the PMC.

NOTE
1 The setting of 0 means that the position switch function of the number is not used.

NOTE
2 Parameters Nos. 6920 to 6925 are valid only when bit 1 (EPW) of parameter No. 6901 is 1.

6930	Maximum value of the operating range of the 1-st position switch
to	
6945	Maximum value of the operating range of the 16-th position switch

[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 diameter 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.
3 The parameter No.6940 to No.6945 is effective when bit 1 (EPW) of parameter No.6901 is set to 1.

6950	Minimum value of the operating range of the 1-st position switch
to	
6965	Minimum value of the operating range of the 16-th position switch

[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 diameter 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.
3 The parameter No.6960 to No.6965 is effective when bit 1 (EPW) of parameter No.6901 is set to 1.

1.3 ERROR COMPENSATION

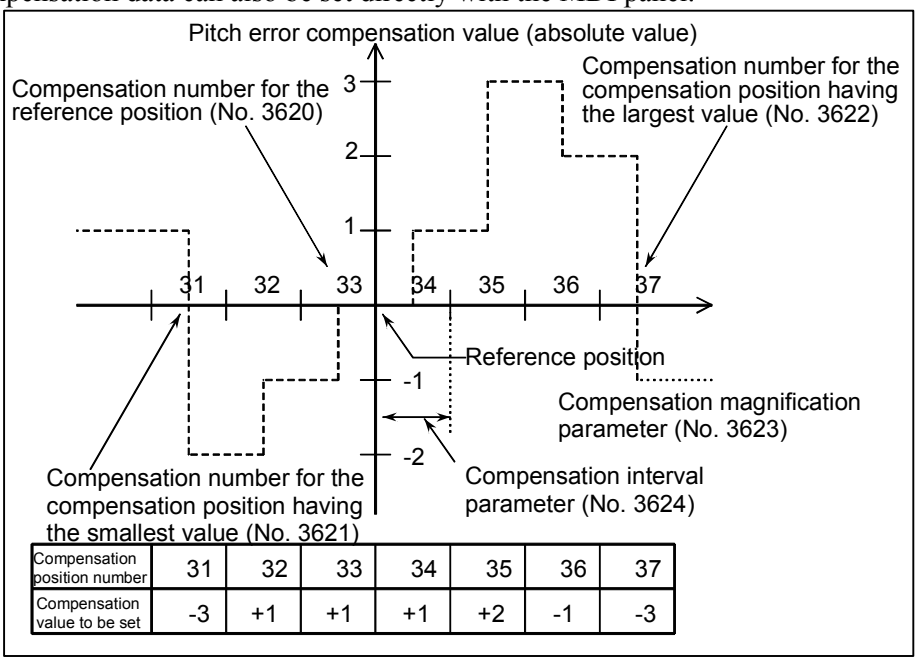
1.3.1 Stored Pitch Error Compensation

Overview

If pitch error compensation data is specified, pitch errors of each axis can be compensated in detection units per axis.

NOTE
To use stored pitch error compensation, set 0 in bit 0 (NPE) of parameter No. 8135.

Pitch error compensation data is set for each compensation position at the intervals specified for each axis. The origin of compensation is the reference position to which the tool is returned.
Pitch error compensation data can be set with external devices such as the Handy File (see Operator's Manual). Compensation data can also be set directly with the MDI panel.



Compensation position number	31	32	33	34	35	36	37
Compensation value to be set	-3	+1	+1	+1	+2	-1	-3

To perform pitch error compensation, it is necessary to set the parameters listed below and specify a pitch error compensation value for the number of the pitch error compensation position set in these parameters. In the above example, "33" is set as the pitch error compensation position number corresponding to the reference point.

- Pitch error compensation position at the reference position (for each axis): Parameter No.3620
- Pitch error compensation position having the smallest value (for each axis): Parameter No.3621
- Pitch error compensation position having the largest value (for each axis): Parameter No.3622
- Pitch error compensation magnification (for each axis): Parameter No.3623
- Interval of the pitch error compensation positions (for each axis): Parameter No.3624

Explanation

- Specifying the compensation position

To assign the compensation positions for each axis, specify the positive direction or the negative direction relative to the compensation position No. of the reference position. If the machine stroke exceeds the specified range on either the positive direction or the negative direction, the pitch error compensation does not apply beyond the range. (All compensation values become 0.)

- Compensation position number

1024 compensation positions from No. 0 to 1023 are available on the pitch error setting screen. Assign arbitrary positions for each axis using parameters.
The number of the compensation position at the reference position (parameter No.3620), number of the compensation position having the smallest value (parameter No.3621), and number of the compensation position having the largest value (parameter No.3622) must be set for each axis.
The name of each axis is displayed before the smallest compensation position number on the pitch error setting screen.

- Interval of compensation positions

The pitch error compensation positions are equally spaced to parameter No. 3624. Set the space between two adjacent positions for each axis.
The minimum interval between pitch error compensation positions is limited and obtained from the following equation:
Minimum interval of pitch error compensation positions = maximum feedrate (rapid traverse rate)/7500
Unit :

Minimum interval of pitch error compensation positions: mm, inches, deg.
Maximum feed rate: mm/min, inch/min, deg/min

Example)

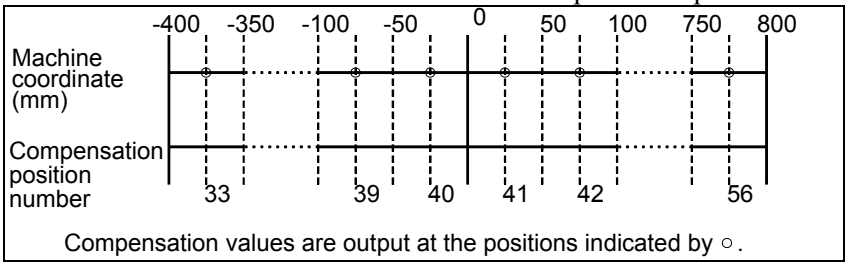
When the maximum rapid traverse rate is 15000 mm/min, the minimum interval between pitch error compensation positions is 2 mm.

Example

- For linear axis

- Machine stroke: -400 mm to +800 mm
- Interval between the pitch error compensation positions: 50 mm
- Number of the compensation position of the reference position: 40

If the above is specified, the number of the farthest compensation position in the negative direction is as follows:
(Number of the compensation position of the reference position) - (Machine stroke on the negative side/Interval between the compensation positions) + 1
= 40 - 400/50 + 1= 33
Number of the farthest compensation position in the positive direction is as follows:
(Number of the compensation position of the reference position) + (Machine stroke on the positive side/Interval between the compensation positions)= 40 + 800/50= 56
The correspondence between the machine coordinate and the compensation position No. is as follows:

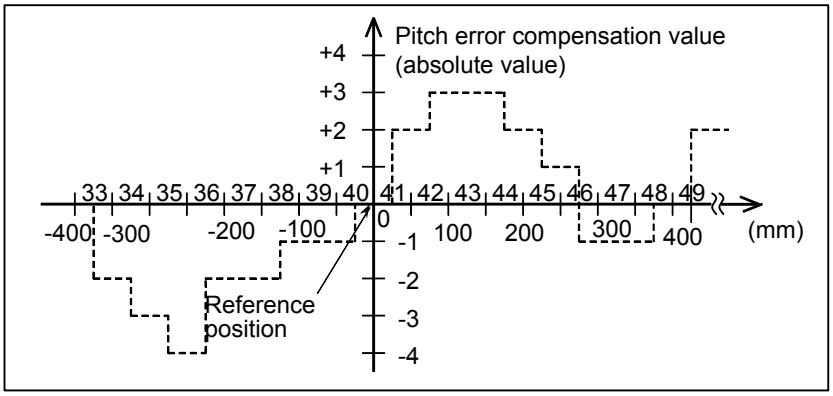


Therefore, set the parameters as follows:

Parameter	Setting value
3620 : Compensation number for the reference position	40
3621 : Smallest compensation position number	33
3622 : Largest compensation position number	56
3623 : Compensation magnification	1
3624 : Interval between pitch error compensation positions	50000

The compensation amount is output at the compensation position No. corresponding to each section between the coordinates.
The following is an example of the compensation amounts.

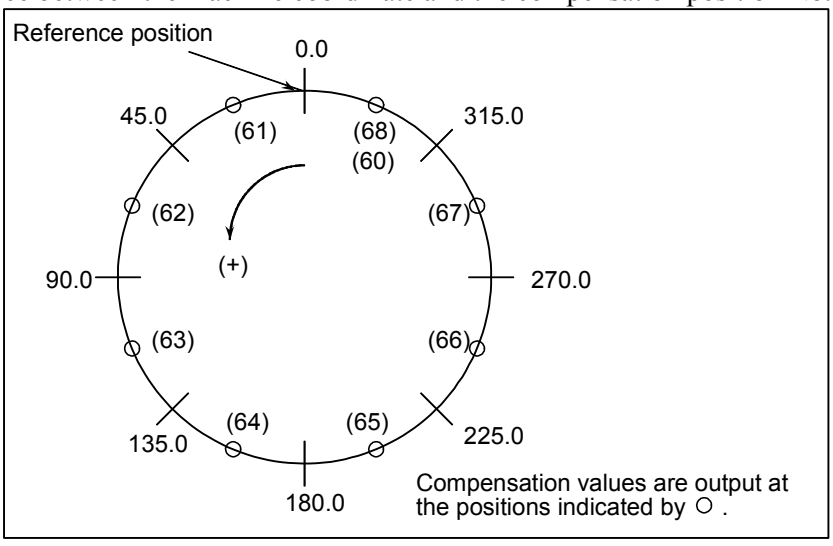
Compensation position No.	33	34	35	36	37	38	39	40	41	42	43	44	45	46	47	48	49		56
Set compensation amount	-2	-1	-1	+2	0	+1	0	+1	+2	+1	0	-1	-1	-2	0	+1	+2		+1



- For rotary axis

- Amount of movement per rotation: 360°
- Interval between pitch error compensation positions: 45°
- Number of the compensation position of the reference position: 60

In the above case, the number of the most distance compensation position on the - side is equal to (the number of the compensation position of the reference position) + 1 = 60 + 1 = 61 for a rotary axis.
The number of the farthest compensation position in the positive direction is as follows:
(Number of the compensation position of the reference position) + (Move amount per rotation/Interval between the compensation positions)= 60 + 360/45= 68
The correspondence between the machine coordinate and the compensation position No. is as follows:



Therefore, set the parameters as follows:

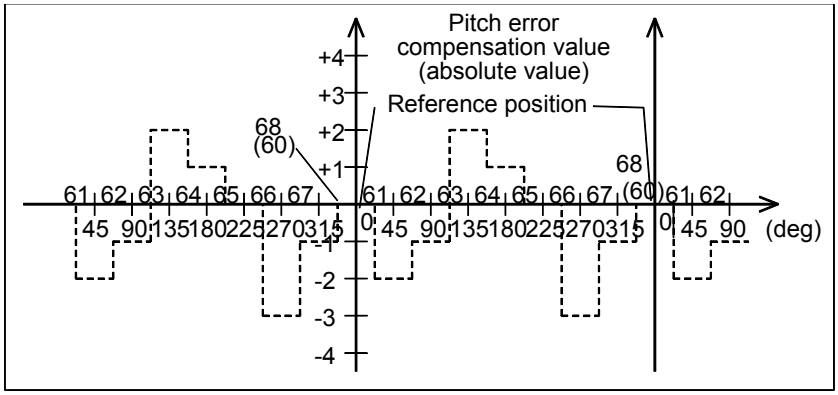
Parameter	Setting value
3620 : Compensation number for the reference position	60
3621 : Smallest compensation position number	61
3622 : Largest compensation position number	68

Parameter	Setting value
3623 : Compensation magnification	1
3624 : Interval between pitch error compensation positions	45000
3625 : Movement value per rotation	360000

If the sum of the compensation values for positions 61 to 68 is not 0, pitch error compensation values are accumulated for each rotation, causing positional deviation.
The same value must be set for compensation positions 60 and 68.

The following is an example of compensation amounts.

Compensation position No.	60	61	62	63	64	65	66	67	68
Set compensation amount	+1	-2	+1	+3	-1	-1	-3	+2	+1



Parameter

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 1023
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 1023
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 1023
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.

	#7	#6	#5	#4	#3	#2	#1	#0
8135								NPE

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 NPE Stored pitch error compensation is:
0: Used.
1: Not Used.

Warning

⚠ WARNING

1 Compensation value range
Compensation values can be set within the range from -7 × compensation magnification (detection unit) to +7 × compensation magnification (detection unit). The compensation magnification can be set for each axis within the range from 0 to 100 in parameter No. 3623.

2 Pitch error compensation of the rotary axis
For the rotating axis, the interval between the pitch error compensation positions shall be set to one per integer of the amount of movement (normally 360°) per rotation. The sum of all pitch error compensation amounts per rotation must be made to 0. Also, set the same compensation value to a position and the same position with one rotation.

⚠ WARNING

3 Conditions where pitch error compensation is not performed

Note that the pitch error is not compensated in the following cases:

- When the machine is not returned to the reference position after turning on the power. This excludes the case where an absolute position detector is employed.
- If the interval between the pitch error compensation positions is 0.
- If the compensation position Nos. on the positive or negative direction do not fall within the range of 0 to 1023.
- If the compensation position Nos. do not conform to the following relationship: Negative side ≤ Reference position < Positive side

Note

NOTE

For 2-path control (T series (2-path control)), axes that have the same axis name but that have different paths must use different compensation position Nos.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Inputting Pitch Error Compensation Data
	Outputting Pitch Error Compensation Data
	Displaying and Setting Pitch Error Compensation Data

1.3.2 Backlash Compensation

Overview

- Backlash compensation

Function for compensating for lost motion on the machine. Set a compensation value in parameter No. 1851, in detection units from 0 to ±9999 pulses for each axis.

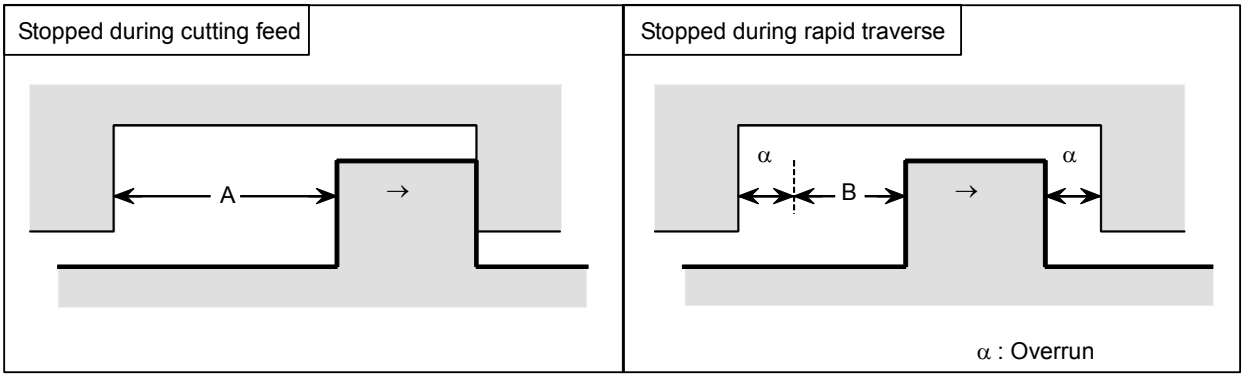
- Backlash compensation for each rapid traverse and cutting feed

More precise machining can be performed by changing the backlash compensating value depending on the feedrate, the rapid traverse or the cutting feed.

Let the measured backlash at cutting feed be A and the measured backlash at rapid traverse be B. The backlash compensating value is shown below depending on the change of feedrate (cutting feed or rapid traverse) and the change of the direction of movement.

Change of feedrate / Change of direction of movement	Cutting feed to cutting feed	Rapid traverse to rapid traverse	Rapid traverse to cutting feed	Cutting feed to rapid traverse
Same direction	0	0	$\pm\alpha$	$\pm(-\alpha)$
Opposite direction	$\pm$A	$\pm$B	$\pm(B+\alpha)$	$\pm(B+\alpha)$

- $\alpha = (A-B) / 2$
- The positive or negative direction for compensating values is the direction of movement.



- Assign the measured backlash at cutting feed (A) in parameter No. 1851 and that at rapid traverse (B) in parameter No. 1852.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1800				RBK				

[Input type] Parameter input
[Data type] Bit path

#4 RBK Backlash compensation applied separately for cutting feed and rapid traverse
0: Not performed
1: Performed

	#7	#6	#5	#4	#3	#2	#1	#0
1802				BKL15				

[Input type] Parameter input
[Data type] Bit axis

#4 BKL15 When the direction of a movement is determined in backlash compensation:
0: The compensation amount is not considered.
1: The compensation amount (pitch error, simple straightness, external machine coordinate system shift, etc.) is considered.

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.

1852	Backlash compensating value used for rapid traverse 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 used in rapid traverse for each axis. (This parameter is valid when RBK, #4 of parameter 1800, is set to 1.) More precise machining can be performed by changing the backlash compensating value depending on the feedrate, the cutting feed or the rapid traverse positioning. Let the measured backlash at cutting feed be A and the measured backlash at rapid traverse be B. The backlash compensating value is shown below depending on the change of feedrate (cutting feed or rapid traverse) and the change of the direction of movement.

Change of feedrate / Change of direction of movement	Cutting feed to cutting feed	Rapid traverse to rapid traverse	Rapid traverse to cutting feed	Cutting feed to rapid traverse
Same direction	0	0	$\pm\alpha$	$\pm(-\alpha)$
Opposite direction	$\pm$A	$\pm$B	$\pm(B+\alpha)$	$\pm(B+\alpha)$

NOTE
1 α=(A-B)/2
2 The positive or negative direction for compensating values is the direction of movement.

Caution

⚠ CAUTION
1 Jog feed is regarded as cutting feed.
2 The backlash compensation for rapid traverse and cutting feed is not performed until the first reference position return is completed after the power is turned on. Under this state, the normal backlash compensation is performed according to the value specified in parameter No. 1851 irrespective of a rapid traverse or a cutting feed.
3 Backlash compensation for each rapid traverse and cutting feed is performed only when 1 is set in bit 4 (RBK) of parameter No. 1800.

1.3.3 Smooth Backlash

NOTE
Smooth backlash function is optional function.

Explanation

With normal backlash compensation, all backlash compensation pulses are output at the location where the direction of axis moving reverses. (Fig. 1.3.3 (a))

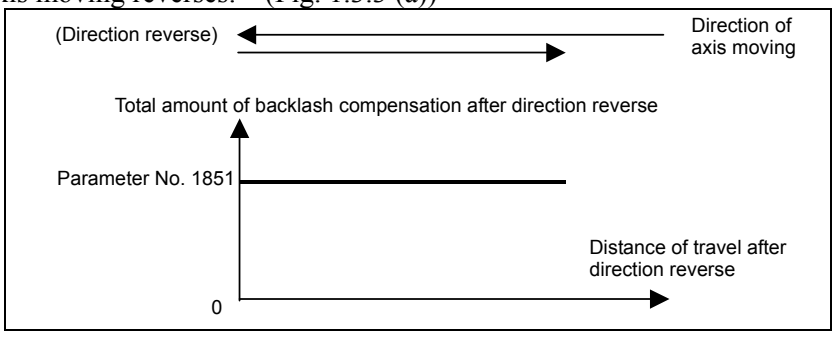


Fig. 1.3.3 (a) Normal backlash compensation

With smooth backlash compensation, backlash compensation pulses are output in accordance with the distance from the location where the direction of axis moving reverses, so that fine backlash compensation corresponding to the characteristics of the machine is possible. (Fig. 1.3.3 (b))

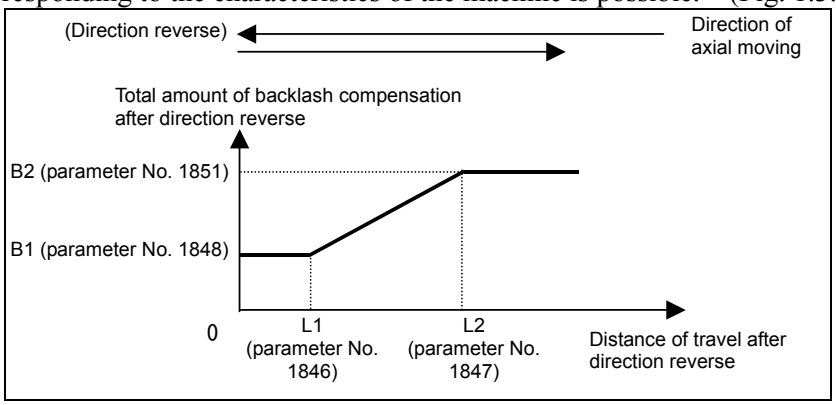


Fig. 1.3.3 (b) Smooth backlash compensation

To enable this function set SBL, bit 2 of parameter No. 1817, to 1.

- First stage backlash compensation output

At the location where the direction of axis moving reverses, the first stage backlash compensation output is performed. Set the first stage backlash compensation B_1, using parameter No. No.1848.

- Second stage backlash compensation output

At the point the tool moves by the distance L1 from the location where the direction of axis moving reverses, the second stage backlash compensation output is started. And, at the point the tool moves by the distance L2 from the location where the direction of axis moving reverses, the second stage backlash compensation output is terminated. The total amount of backlash at the stage where the second stage backlash compensation output is terminated, or B2, is the same as the backlash compensation set using parameter No. No.1851. Set the distances L1 and L2, using parameters Nos. 1846 and 1847, respectively.

If backlash compensation for each rapid traverse and cutting feed is enabled (RBK, bit 4 of parameter No. 1800 = 1), the total amount of backlash compensation at the stage where the second stage backlash compensation output is terminated, or B2, is the backlash compensation as determined by parameters Nos. 1852 and 1851, the reversed direction, and the rapid traverse/cutting feed mode. The rate of increase of the second stage backlash compensation output remains the same as that during cutting. (Expression 1)

$$Rate\,of\,increase\,of\,\sec ond\,backlash\,compensation\,output = \frac{Parameter\,\mathrm{No.}1851 - \mathrm{B}_1}{\mathrm{L}_2 - \mathrm{L}_1} \qquad (1)$$

The following shows an example in which the tool is changed from cutting feed to rapid traverse feed and the direction reverses. (Fig. 1.3.3 (c))

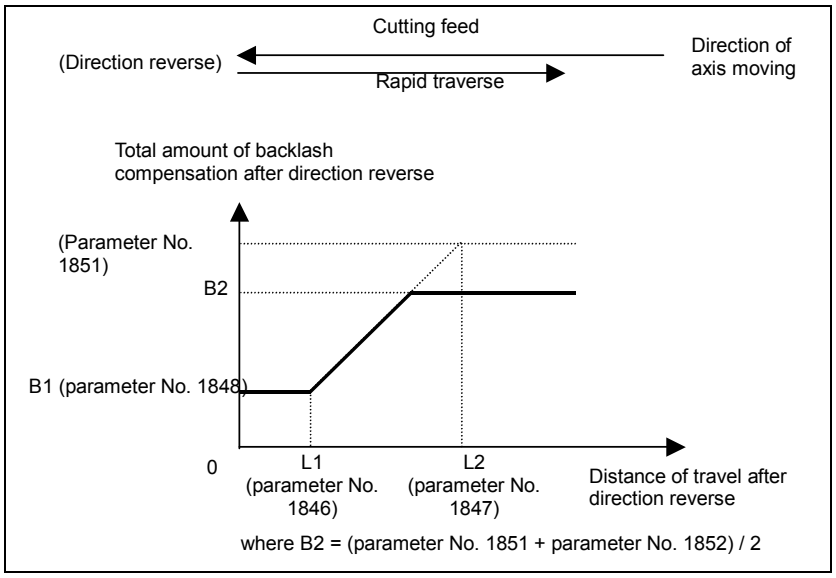


Fig. 1.3.3 (c) In the case of a change from cutting feed to rapid traverse

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1817						SBL		

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#2 SBL Smooth backlash compensation is :
0: Disabled.
1: Enabled.

NOTE
Smooth backlash compensation function is optional function.

1846	Distance for starting the second stage of smooth backlash compensation

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999
For each axis, set the distance from the point where the axis movement direction is reversed to the point where the second stage of smooth backlash compensation is started.

If the following condition is not satisfied, smooth backlash compensation is disabled:
Value of parameter No. 1846 ≥ 0
Value of parameter No. 1846 < value of parameter No. 1847

1847	Distance for ending the second stage of smooth backlash compensation

[Input type] Parameter input
[Data type] 2-word axis

[Unit of data] Detection unit
[Valid data range] 0 to 999999999
For each axis, set the distance from the point where the axis movement direction is reversed to the point where the second stage of smooth backlash compensation is ended.

If the following condition is not satisfied, smooth backlash compensation is disabled:
Value of parameter No. 1846 ≥ 0
Value of parameter No. 1846 < value of parameter No. 1847

1848	Value of the first stage of smooth backlash compensation

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] -9999 to 9999
Set the value of the first stage of smooth backlash compensation for each axis.
If the setting of this parameter is greater than the total amount of backlash compensation, smooth backlash compensation is not performed.
If the backlash compensating value (No. 1851) for each axis is negative, set this parameter to a negative value. If the sign of the backlash compensating value (No. 1851) for each axis is different, perform compensation with the value of the first stage of smooth backlash compensation assumed to be 0.

1.3.4 Simple Straightness Compensation (M Series)

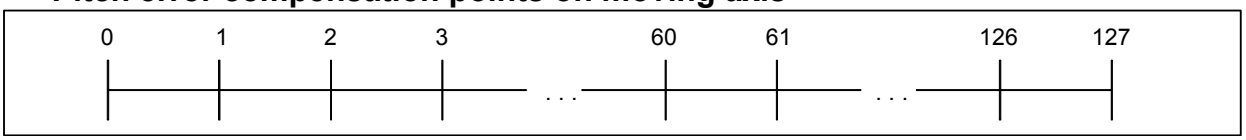
M

Overview

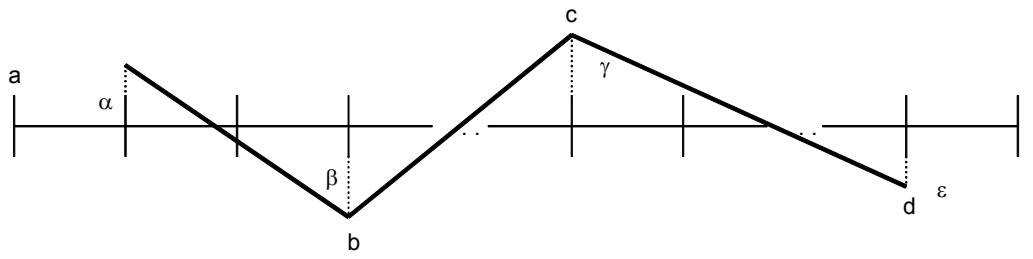
For a machine tool with a long stroke, deviations in straightness between axes may affect the machining accuracy. For this reason, when an axis moves, other axes are compensated in detection units to improve straightness. This improvement results in better machining accuracy.
When an axis (parameter No. 5711) moves, the corresponding compensation axis (parameter No. 5721) is compensated.
That is, the compensation axis is compensated at the pitch error compensation position (See Subsection "Stored Pitch Error Compensation") of the moving axis.

Example

- **Pitch error compensation points on moving axis**



- Machine coordinates for moving axis



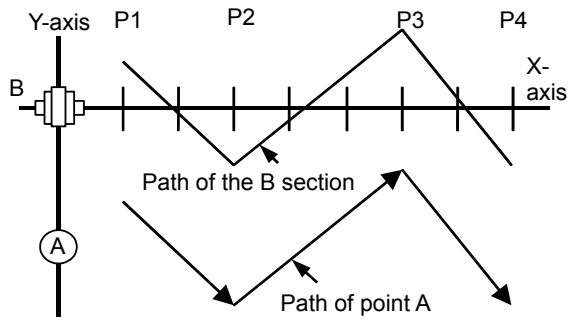
a, b, c, d : Compensation position numbers of the moving axis (This number is originally a pitch error compensation position number.)
α, β, γ, ε : Compensation amount for the compensation position number

The compensation from point a to point b is calculated from the formula: (β-α)/(b-a).

NOTE
The simple straightness compensation function (M series) is optional.
To use the simple straightness compensation function, enable the stored pitch error compensation function (set 0 in bit 0 (NPE) of parameter No. 8135).

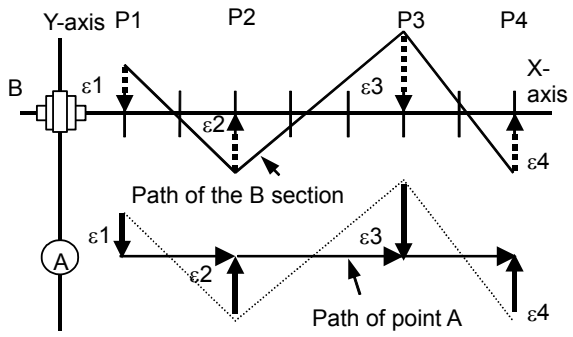
- Example

Imagine a table whose Y-axis ball screw is placed on its X-axis ball screw. If the X-axis ball screw is inclined at a certain angle because of, for example, bending, the machining precision related to the Y-axis becomes low because its ball screw is affected by the gradient of the X-axis ball screw. (Left figure shown below)
Specifying the X-axis and Y-axis, respectively, as a moving axis and a compensation axis by means of simple straightness compensation causes the Y-axis (compensation axis) position to be compensated according to the X-axis (moving axis) position, thus increasing the machining precision. (Right figure shown below)



P1, P2, P3, P4 : Points on the moving axis

The path of the B section, which is a joint between the X-axis and Y-axis, is affected by the gradient of the X-axis because of the structure of the table. If a command specifies movement from P1 to P4 only along the X-axis without applying straightness compensation, the path of point A along the Y-axis is affected by the gradient of the X-axis.



P1, P2, P3, P4 : Points on the moving axis
ε1, ε2, ε3, ε4 : Compensation amount for each compensation point along the compensation axis

If a command specifies movement from P1 to P4 only along the X-axis (moving axis), when the B section moves in the sequence P1 → P2 → P3 → P4, straightness compensation gives compensation amounts ε1 to ε4 to the Y-axis (compensation axis). This Y-axis compensation keeps the path of point A along the Y-axis from being affected by the gradient of the X-axis even when the B section, which is a joint between the X-axis and Y-axis, is affected by the gradient of the X-axis.

Parameter

5711	
	Simple straightness compensation : Axis number of moving axis 1

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
Set the axis number of a moving axis in simple straightness compensation.
When 0 is set, compensation is not performed.

5721	
	Simple straightness compensation : Axis number of compensation axis 1 for moving axis 1

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
Set the axis number of a compensation axis in simple straightness compensation.
When 0 is set, compensation is not performed.

5731	
	Simple straightness compensation : Compensation point number a of moving axis 1
to	to
5734	
	Simple straightness compensation : Compensation point number d of moving axis 1

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Unit of data] Detection unit
[Valid data range] 0 to 1023
These parameters set compensation point numbers in stored pitch error compensation.
Set four compensation points for each moving axis.

5761	Compensation corresponding compensation point number a of moving axis 1
to	to
5764	Compensation corresponding compensation point number d of moving axis 1

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Word path
[Unit of data] Detection unit
[Valid data range] -32767 to 32767
Each of these parameters sets a compensation value for each moving axis compensation point.

Alarm and message

Number	Message	Description
PW5046	ILLEGAL PARAMETER (S-COMP.)	The parameter for setting simple straightness compensation is incorrect.

Note

NOTE
1 The simple straightness compensation function can be used after a moving axis and its compensation axis have returned to the reference position.
2 After setting parameters for simple straightness compensation, be sure to turn off the NC power.
3 Set parameters for simple straightness compensation according to the following conditions:
- The compensation at a compensation position must be within the range -127 to 127.
- Compensation positions must be set so that "$a \leq b \leq c \leq d$" is satisfied.
- Compensation positions must exist between the compensation position with the largest positive value and that with the largest negative value in the stored pitch error compensation data for each axis. Four compensation positions can be set to 0 at a time. In this case, compensation is not performed.

4 To add the option of the simple straightness compensation function, enable the stored pitch error compensation function (set 0 in bit 0 (NPE) of parameter No. 8135).
In this case, the number of compensation positions of each axis between the compensation position with the largest positive value and that with the largest negative value in the stored pitch error compensation data must be equal to or less than 1536.
5 Simple straightness compensation data is superposed on stored pitch error compensation data and output. Straightness compensation is performed at pitch error compensation intervals.
6 Simple straightness compensation does not allow the moving axis to be used as a compensation axis. To implement such compensation, use gradient compensation (see Subsection "Gradient Compensation").

1.3.5 Gradient Compensation

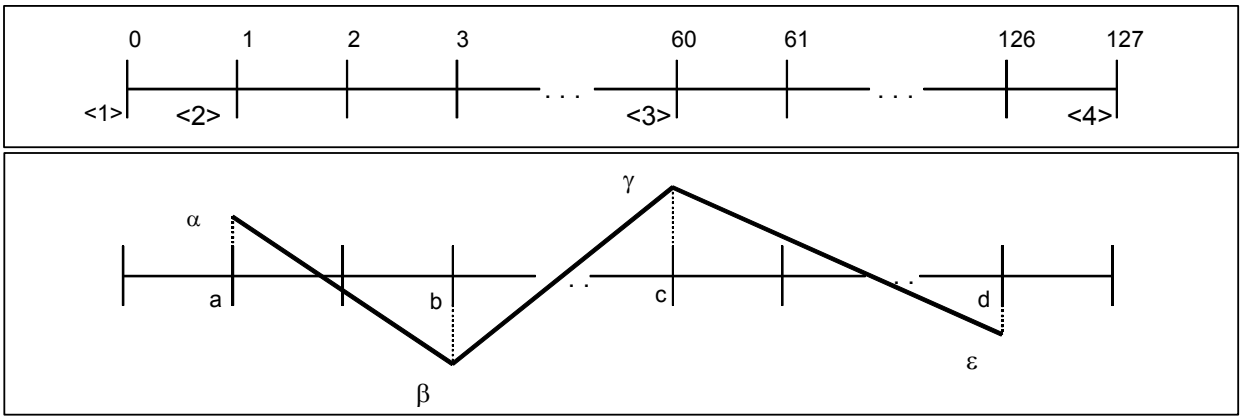
Overview

By compensating for those errors in tools such as feed screws that depend on the position of the machine system in detection units, machining precision can be improved and mechanical life can be prolonged. Compensation is performed along an approximate straight line formed with a parameter-specified compensation point and a compensation amount related to it.

NOTE
The gradient compensation function is optional function.
To use the gradient compensation function, enable the stored pitch error compensation function (set 0 in bit 0 (NPE) of parameter No. 8135).

Explanation

Three approximate straight lines are formed with four parameter-specified compensation points and compensation amounts related to the respective compensation points. Gradient compensation is carried out along these approximate straight lines at pitch error compensation intervals. The gradient compensation amount is added to the pitch error compensation amount.



To perform gradient compensation, stored pitch error compensation must be set for the axis subject to compensation.

(1) Number of the most distant pitch error compensation point on the - side (parameter No.3621)
(2) Pitch error compensation point interval (parameter No. 3624)
(3) Number of the pitch error compensation point of the reference position (parameter No. 3620)
(4) Number of the most distant pitch error compensation point on the + side (parameter No. 3622)

Gradient compensation parameters must be set.

a,b,c,d : Compensation point numbers. (Pitch error compensation point numbers are used. Parameters Nos. 5861 to 5864)

$\alpha,\beta,\gamma,\varepsilon$: Compensation amounts at compensation points a, b, c, and d (parameters Nos. 5871 to 5874)

In above figure, a, b, c, and d are 1, 3, 60, and 126, respectively.

Unlike stored pitch error compensation, whose amount is set up for an individual compensation point, an amount of gradient compensation is calculated for individual compensation points by setting up four typical points and compensation amounts for them.

Example:

In above figure, the compensation amounts at the individual compensation points located between points a and b are $(\beta-\alpha)/(b-a)$.

Parameter

5861	Inclination compensation : Compensation point number a for each axis
5862	Inclination compensation : Compensation point number b for each axis
5863	Inclination compensation : Compensation point number c for each axis
5864	Inclination compensation : Compensation point number d for each axis

NOTE
When these parameters are 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
These parameters set the compensation points for inclination compensation. The points are set for the compensation point numbers for stored pitch error compensation.

5871	Inclination compensation : Compensation α at compensation point number a for each axis
5872	Inclination compensation : Compensation β at compensation point number b for each axis
5873	Inclination compensation : Compensation γ at compensation point number c for each axis
5874	Inclination compensation : Compensation δ at compensation point number d for each axis

NOTE
When these parameters are 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] -32767 to 32767
Each of these parameters sets a compensation value for each axis compensation point.

Alarm and message

Number	Message	Description
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 1023 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.

Note

NOTE

1 Gradient compensation is enabled after the reference position is established on the compensation axis.
2 When setting parameters Nos. 5861 to 5864 (compensation point numbers a to d for individual axes), turn off the power to the NC and then back ON for the settings to take effect.
3 During automatic operation, it is possible to overwrite parameters Nos. 5871 to 5874, but make sure that all axes are stopped beforehand. If any of parameters Nos. 5871 to 5874 (compensation amounts at compensation points a to d for individual axes) is changed, the compensation amount determined from the compensation amount after the change is output after the point at which to output the compensation amount for the next gradient compensation is passed.
4 Parameters must satisfy the following conditions:
  - The compensation amount at a compensation point must be in the range of -127 to 127.
  - Compensation points must satisfy the following relationships: $a \leq b \leq c \leq d$.
  - Compensation points must be located between the most distant compensation point in stored pitch error compensation on the - side of each axis and the most distant compensation point on the + side. If all four points are equal to 0, compensation is not performed.
5 To use the gradient compensation function, enable the stored pitch error compensation function (set 0 in bit 0 (NPE) of parameter No. 8135).
  The number of compensation points located between the most distant compensation point on the + side of each axis and the most distant compensation point on the + side in stored pitch error compensation must not exceed 1023.
6 Gradient compensation is superimposed on the stored pitch error compensation data.
7 This function is applied to both linear and rotation axes.
8 The compensation amount at the reference position is output based on parameter settings. The first compensation pulse is output when the compensation point is reached.

Warning

⚠ WARNING

If any of parameters Nos. 5871 to 5874 (compensation amounts at compensation points a to d for individual axes) is changed, very large compensation may be output depending on the setting. Great care should be taken.

1.3.6 Bi-directional Pitch Error Compensation

Overview

In bi-directional pitch error compensation, different pitch error compensation amounts can be set for travel in the positive direction and that in the negative direction, so that pitch error compensation can be performed differently in the two directions, in contrast to stored pitch error compensation, which does not distinguish between the directions of travel. In addition, when the direction of travel is reversed, the compensation amount is automatically calculated from the pitch error compensation data to perform compensation in the same way as in backlash compensation. This reduces the difference between the paths in the positive and negative directions.

NOTE
Bi-directional pitch error compensation function is optional function.
To use the bi-directional pitch error compensation function, enable the stored pitch error compensation function (set 0 in bit 0 (NPE) of parameter No. 8135). Note also that this function cannot be used with an axis for which gradient compensation is to be performed.

Explanation

- Setting data

1. Setting parameters
   Set the following parameters for each axis.

Table 1.3.6 (a)

Parameter number	Description
3605#0	Bidirectional pitch error compensation, 1: Enabled / 0: Disabled
3620	Number of the pitch error compensation point of the reference position
3621	Number of the most distant pitch error compensation point on the - side for travel in the positive direction
3622	Number of the most distant pitch error compensation point on the + side for travel in the positive direction
3623	Pitch error compensation magnification
3624	Pitch error compensation point interval
3625	For a rotation axis, amount of travel per rotation in pitch error compensation
3626	Number of the most distant pitch error compensation point on the - side for travel in the negative direction
3627	Pitch error compensation amount (absolute value) at the reference position when the machine moves to the reference position in the direction opposite to that of a reference position return

2. Pitch error compensation data
   The compensation point numbers can be from 0 to 1023 and from 3000 to 4023. This data may be used for both the positive and negative directions. Note, however, that the set of compensation data for a given axis cannot extend over 1023 and 3000.

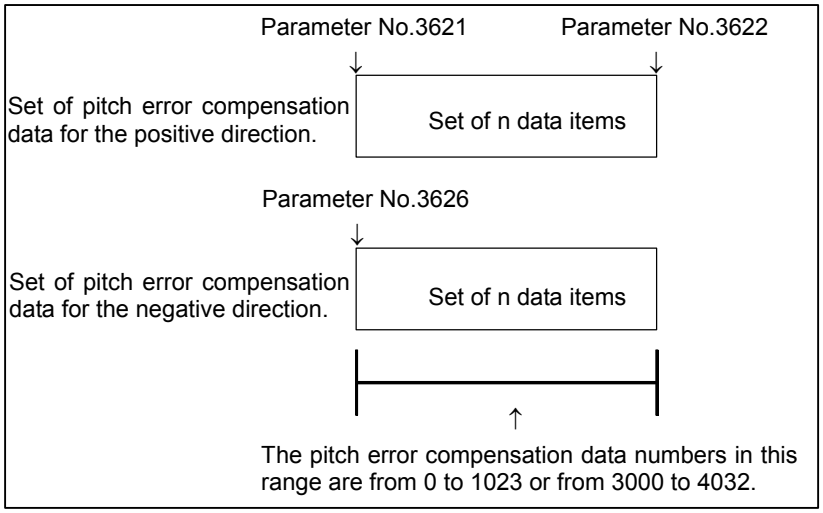


Fig. 1.3.6 (a)

- Data setting example

If the direction of a manual reference position return is positive on an axis (linear axis) having the pitch error amounts shown in the figure below (Fig. 1.3.6 (b)), set the data given in the table below (Table 1.3.6 (b)).

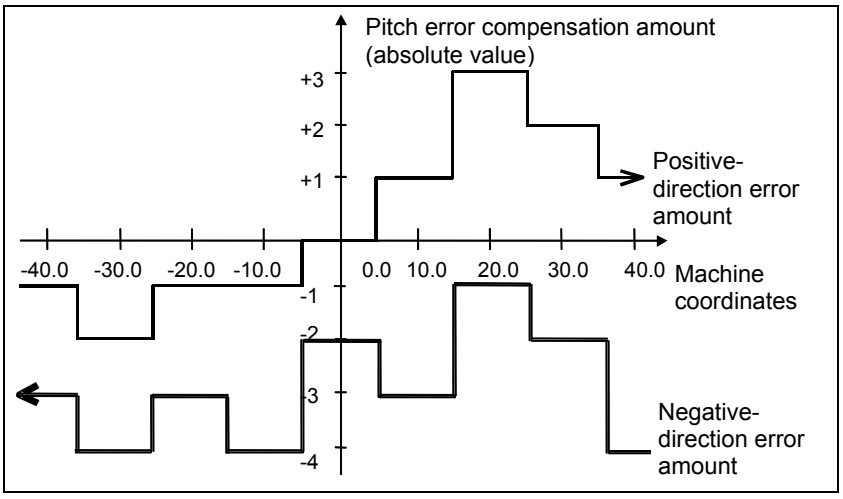


Fig. 1.3.6 (b)

Table 1.3.6 (b) Positive-direction pitch error data

Compensation point number	20	21	22	23	24	25	26	27
Compensation amount to be set	-1	+1	0	+1	+1	+2	-1	-1

As pitch error data, always set incremental values as viewed in the negative direction (direction toward the left in Fig. 1.3.6 (b)).

Table 1.3.6 (c) Negative-direction pitch error data

Compensation point number	30	31	32	33	34	35	36	37
Compensation amount to be set	-1	+1	-1	+2	-1	+2	-1	-2

Set negative-direction pitch error data for all the points for which positive-direction pitch error data has been set.
As negative-direction pitch error data, always set incremental values as viewed in the negative direction.

Table 1.3.6 (d)

Parameter number	Setting	Description
3605#0	1	Bi-directional pitch error compensation, 1: Enabled / 0: Disabled

Parameter number	Setting	Description
3620	23	Number of the pitch error compensation point for the reference position
3621	20	Number of the most distant pitch error compensation point on the - side for travel in the positive direction
3622	27	Number of the most distant pitch error compensation point on the + side for travel in the positive direction
3623	1	Pitch error compensation magnification
3624	10000	Pitch error compensation point interval
3625	-	For a rotation axis, amount of travel per rotation in pitch error compensation
3626	30	Number of the most distant pitch error compensation point on the - side for travel in the negative direction
3627	-2	Pitch error compensation amount (absolute value) at the reference position when the machine moves to the reference position in the direction opposite to that of the reference position return

This example assumes that the direction of a manual reference position return is positive. For parameter No. 3627, therefore, set -2, which is the pitch error compensation amount (absolute value) at the reference position when the machine moves to the reference position in the negative direction.

- Compensation example

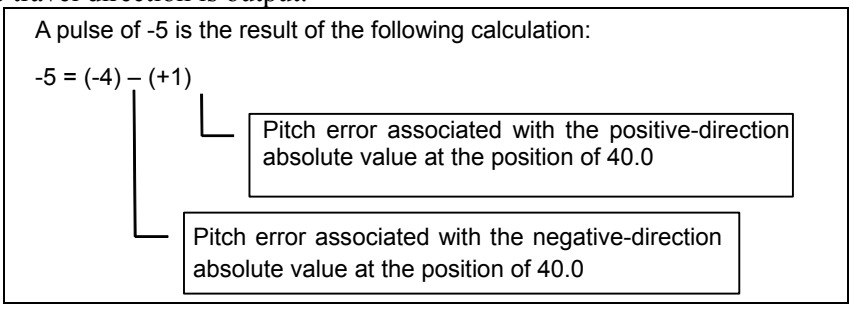
If, in the setting example given in the previous section, the machine moves
0.0 to 40.0,
40.0 to -40.0, and
-40.0 to 0.0
for a manual reference position return, pitch error compensation pulses are output as follows:

Machine coordinate	0.0	5.0	15.0	25.0	35.0	40.0
Compensation pulse	-	+1	+2	-1	-1	-5

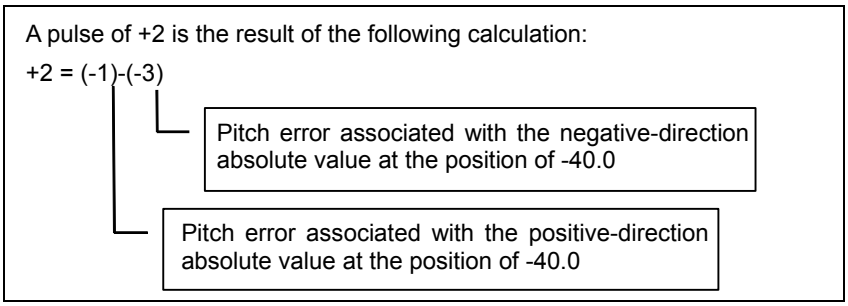
Machine coordinate	35.0	25.0	15.0	5.0	-5.0	-15.0	-25.0	-35.0	-40.0
Compensation pulse	+2	+1	-2	+1	-2	+1	-1	+1	+2

Machine coordinate	-35.0	-25.0	-15.0	-5.0	0.0
Compensation pulse	-1	+1	0	+1	-

When the travel direction changes from positive to negative at the position of 40.0, the compensation for the reverse of the travel direction is output.



When the travel direction changes from negative to positive at the position of -40.0, the compensation for the reverse of the travel direction is output.



- Setting and displaying data

All the compensation data can be displayed and set on the conventional screen for the pitch error compensation data.
And those data can be input and output by the following methods.

- Input by MDI
- Input by G10
- Input and output by input/output device interface
- Input by PMC window (function code 18)

(It is not possible to input and output by the method other than the above methods.)
Output format : The output format is as follows:

N20000 P.... ;
N21023 P.... ;
N23000 P.... ;
N24023 P.... ;
N : Pitch error compensation point No. + 20000
P : Pitch error compensation data

Parameter

	#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 1023
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 1023
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 (when moving in the positive 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
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.

NOTE
2 A set of compensation data items for a single axis should not be set to lie astride 1023 and 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.

Note

NOTE
1 To use this function, enable the stored pitch error compensation function (set 0 in bit 0 (NPE) of parameter No. 8135).
2 This function is enabled after a manual reference position return or an automatic reference position return with the same sequence as that of a manual reference position return is performed. When an absolute position detector is used, however, the function is enabled after the power is turned on.
3 When the machine moves to the reference position in the reference position return direction, set the absolute value of the pitch error compensation pulse to 0.
4 When this function and backlash compensation are used at the same time, the pulse resulting from backlash compensation is superimposed on the compensation pulse when the travel direction is reversed.
5 When this function is used for a rotation axis, the sum of the pitch error compensation amounts per rotation about the rotation axis must be 0 for both the positive and negative directions.
6 The function cannot be used with the inclination compensation function. (This function cannot be used with an axis for which gradient compensation is to be performed.)

1.3.7 Differences between Pitch Error Compensation, Simple Straightness Compensation, and Gradient Compensation (Reference)

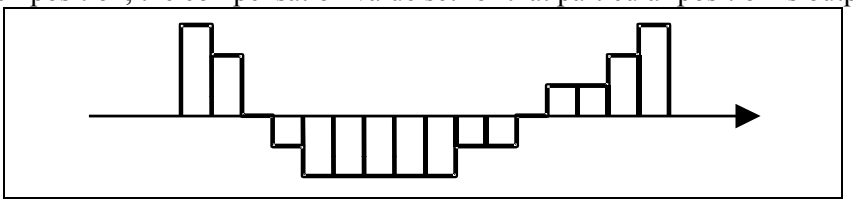
Overview

The pitch error compensation, simple straightness compensation, and gradient compensation functions all divide a machine stroke at parameter-set compensation intervals and perform compensation at each compensation position based on the machine position.
Gradient compensation and simple straightness compensation use the compensation interval and compensation position numbers set for pitch error compensation. As for the compensation amount, however, specific values are provided that are respectively dedicated to each compensation function including gradient compensation and simple straightness compensation.

Explanation

- Pitch error compensation

In the case of pitch error compensation, a compensation value is set for each compensation position. At each compensation position, the compensation value set for that particular position is output.

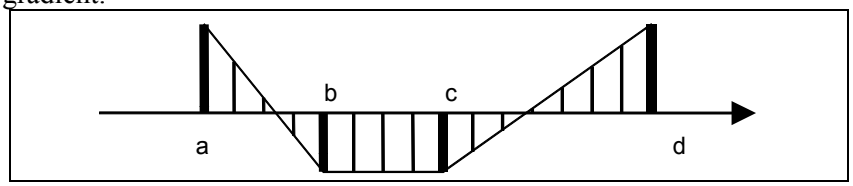


- Bi-directional pitch error compensation

In the case of bi-directional pitch error compensation, the compensation value can be changed depending on the direction of axis movement.

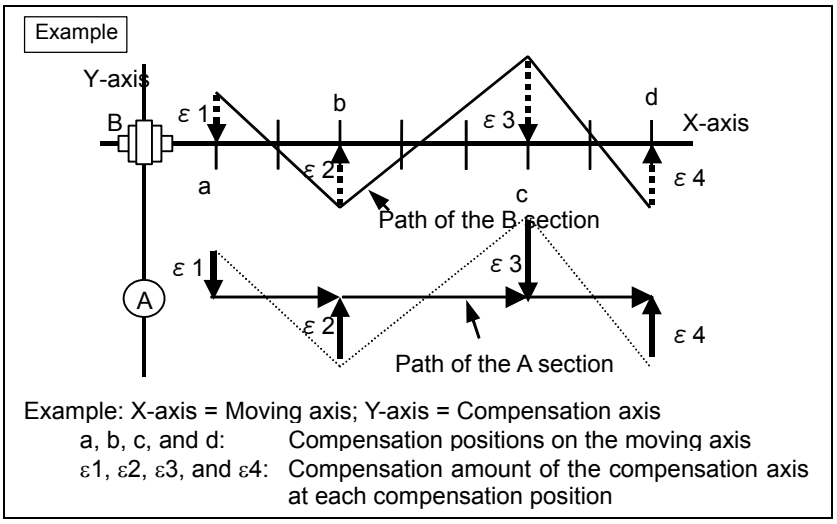
- Gradient compensation

In the case of gradient compensation, four representative positions (a, b, c, and d) are selected from the compensation positions set for pitch error compensation (compensation positions for gradient compensation), inseatd of setting a compensation value for each compensation position. Compensation values are set only for those four positions. At each compensation position for pitch error compensation that is present between the compensation positions for gradient compensation, the NC calculates and outputs a compensation value as appropriate for gradient compensation. This can be applied when the pitch error has a gradient.



- Simple straightness compensation

In the case of simple straightness compensation, four representative positions (a, b, c, and d) are selected from the compensation positions set for pitch error compensation (compensation positions for simple straightness compensation), as with gradient compensation. Compensation values are set only for those four positions. At each compensation position for pitch error compensation that is present between the compensation positions for gradient compensation, the NC calculates and outputs a compensation value as appropriate for simple straightness compensation.
One significant difference from gradient compensation, in which the moving axis is adjusted, is that the moving axis and the compensation axis are different. This relationship needs to be set using the relevant parameters (e.g., the Y-axis is to be adjusted as the X-axis is moved).



Example: X-axis = Moving axis; Y-axis = Compensation axis
a, b, c, and d: Compensation positions on the moving axis
ε1, ε2, ε3, and ε4: Compensation amount of the compensation axis at each compensation position

1.4 SETTINGS RELATED TO SERVO-CONTROLLED AXES

The servo interface of the Series 16 features the following:

- Digitally controlled AC servo motor
- Motor feedback with serial pulse coders
  (1) Absolute pulse coder with a resolution of 1,000,000 pulses/rev
  (2) Absolute pulse coder with a resolution of 65,536 pulses/rev
  (3) Incremental pulse coder with a resolution of 10,000 pulses/rev
- Scale feedback with A/B/Z signal interface

1.4.1 Parameters Related to Servo

Overview

Terms frequently used in explanation of parameters related to servo systems are listed below:

Least command increment
The minimum unit of a command to be given from CNC to the machine tool

Detection unit
The minimum unit which can detect the machine tool position

Command multiplier (CMR)
A constant to enable the weight of CNC command pulses to meet the weight of pulses from the detector

Detection multiplier (DMR)
A constant to enable the weight of CNC command pulses to meet the weight of pulses from the detector

⚠ CAUTION
The relations among the least command increment, detection unit, CMR, and DMR are as specified below.
Detection unit =
Move amount per revolution of motor /
DMR × number of pulses of detector per revolution
Least command increment = CMR × detection unit
The flexible feed gear function in the digital servo defines constant DMR using two parameters (Nos. 2084 and 2085) n and m (DMR = n/m).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1800							CVR	

[Input type] Parameter input
[Data type] Bit path

#1 CVR When velocity control ready signal VRDY is set ON before position control ready signal PRDY comes ON
0: A servo alarm is generated.
1: A servo alarm is not generated.

	#7	#6	#5	#4	#3	#2	#1	#0
1815			APCx	APZx			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 pulse coder is not used.
1: A separate pulse coder 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).

#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 pulse coder)

	#7	#6	#5	#4	#3	#2	#1	#0
1816		DM3x	DM2x	DM1x				

[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 DM1x
#5 DM2x
#6 DM3x

By using DM1x, DM2x, and DM3x, a detection multiplication factor (DMR) is set.
This parameter is valid when a separate position detector (AB phase) is used and parameter No. 2084 and No. 2085 are not set.

DM3x	DM2x	DM1x	DMR
0	0	0	1/2
0	0	1	1
0	1	0	3/2
0	1	1	2
1	0	0	5/2
1	0	1	3
1	1	0	7/2
1	1	1	4

NOTE
For the FS0*i*-C, one of the following changes is required besides setting bit 3 (DIAx) of parameter No. 1006 so that the axis based on diameter specification achieves the specified amount of movement.
- Halve the command multiplication (the detection unit is not changed).
- Halve the detection unit and double the flexible feed gear (DMR).

For the FS0*i*-D, only if bit 3 (DIAx) of parameter No. 1006 is set, the CNC halves the specified pulse. Accordingly, the above changes are not required (when the detection unit is not changed). To halve the detection unit, double both CMR and DMR.

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

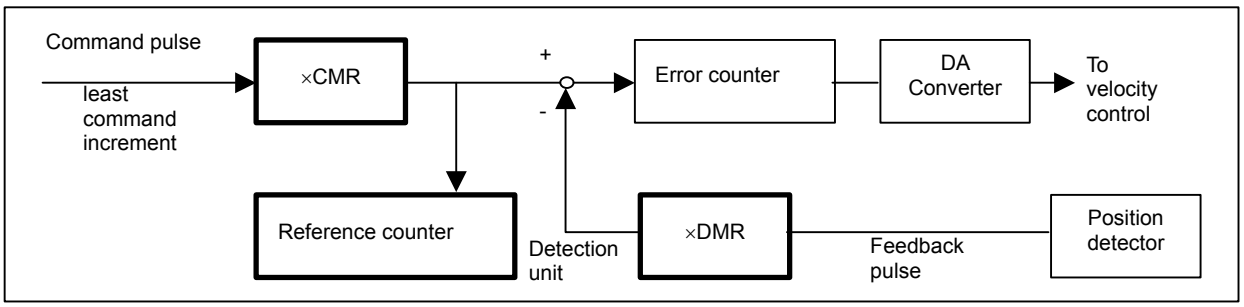
			Least input increment	Least command increment
IS-B	Millimeter machine	Millimeter input	0.001 mm (diameter specification)	0.0005 mm
			0.001 mm (radius specification)	0.001 mm
		Inch input	0.0001 inch (diameter specification)	0.0005 mm
			0.0001 inch (radius specification)	0.001 mm
	Inch machine	Millimeter 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
	Rotation 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
	Rotation axis		0.0001 deg	0.0001 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
Rotation 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 pulse coder]/[Number of pulses per rotation of the pulse coder]

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 pulse coder]

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
1 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
2 For the FS0*i*-C, one of the following changes is required besides setting bit 3 (DIAx) of parameter No. 1006 so that the axis based on diameter specification achieves the specified amount of movement.
- Halve the command multiplication (the detection unit is not changed).
- Halve the detection unit and double the flexible feed gear (DMR).

For the FS0*i*-D, only if bit 3 (DIAx) of parameter No. 1006 is set, the CNC halves the specified pulse. Accordingly, the above changes are not required (when the detection unit is not changed). To halve the detection unit, double both CMR and DMR.

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.

1825	Servo loop gain for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 0.01/sec
[Valid data range] 1 to 9999

Set the loop gain for position control for each axis.
When the machine performs linear and circular interpolation (cutting), the same value must be set for all axes. When the machine requires positioning only, the values set for the axes may differ from one another. As the loop gain increases, the response by position control is improved. A too large loop gain, however, makes the servo system unstable.
The relationship between the positioning deviation (the number of pulses counted by the error counter) and the feedrate is expressed as follows:

Positioning deviation = Feedrate / (60 × Loop gain)

Unit : Positioning deviation mm, inch or deg
Feedrate mm/min, inch/min, or deg/min
Loop gain 1/sec

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) 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) is generated, and operation is stopped immediately (as in emergency stop).

1850	Grid shift and reference position shift 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] -99999999 to 99999999

To shift the reference position, the grid can be shifted by the amount set in this parameter. Up to the maximum value counted by the reference counter can be specified as the grid shift.

In case of parameter SFDx(No.1008#4) is 0: Grid shift

In case of parameter SFDx(No.1008#4) is 1: Reference point shift

NOTE
For setting the reference position without dogs, only the grid shift function can be used.
(The reference position shift function cannot be used.)

1.4.2 Absolute Position Detection

Overview

An absolute position detector (absolute pulse coder) is an incremental pulse coder with an absolute counter. It detects the absolute position based on the value of the absolute counter. For an axis on which an absolute position detector is mounted, no reference position return is required at power-on because the machine position is always stored with batteries if the power to the CNC is turned off.

When the machine position has been brought into correspondence with the absolute position detector, the current position is read from the absolute counter at CNC power on and the machine and workpiece coordinate systems are automatically set using the value. In this case, you can immediately start automatic operation.

Restrictions described in the OPERATOR'S MANUAL and others that include those listed below are removed:

- "Reference position return is required after power-on."
- "The CNC can be used after reference position return is performed after power-on."

Explanation

- Coordinate systems at power-on

(1) The machine and workpiece coordinate systems are automatically set. When bit 3 (PPD) of parameter No. 3104 is set to 1, the relative position display is preset.

(2) The amount of shift by coordinate system setting (G92 for the M series or G50 for the T series) or local coordinate system setting (G52) is cleared.

- Setting the zero point of an absolute position detector

In the following cases, set the zero point of an absolute position detector (bring the counter value of the absolute position detector into correspondence with the reference position):

- Primary adjustment is performed (after the reference position is determined).
- The reference position is changed.
- An absolute position detector is replaced with another.
- A servo motor is replaced with another.
- Alarm DS0300 is issued.
- The file memory is replaced with another.
- Parameter data is cleared.

To set the zero point of an absolute position detector, the following two methods are available: One method using manual reference position return and the other using MDI operation.

- Manual reference position return

Follow the procedure below to perform manual reference position return.

(1) Set bit 4 (APZ) of parameter No. 1815 to 0. Alarms (PW0000) and (DS0300) are displayed.

(2) Turn the power off, then on again. Alarm (DS0300) is displayed.
(3) Perform manual reference position return. When manual reference position return is completed, bit 4 (APZ) of parameter No. 1815 is automatically set to 1.
(4) Press the reset button to release alarm DS0300.

Signal

Absolute position detector battery voltage low alarm signal PBATL<Fn172.7>

[Classification] Output signal
[Function] Notifies that the life of the absolute position detector battery is about to expire.
[Operation] These signal is 1 in the following case:
- The battery voltage for the absolute position detector becomes lower than or equal to the rating.
  The battery need be replaced in the immediate future.

These signal is 0 in the following case:
- The battery voltage for the absolute position detector is higher than or equal to the rating.

Absolute position detector battery voltage zero alarm signal PBATZ<Fn172.6>

[Classification] Output signal
[Function] Notifies that the life of the absolute position detector battery has expired.
[Operation] These signals are 1 in the following case:
- The batteries for the absolute position detector have run out. The batteries need be replaced in the status in which the power to the machine is on.

These signals are 0 in the following case:
- The batteries for the absolute position detector feed the voltage required for data backup.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn172	PBATL	PBATZ						

Diagnosis display

Why bit 4 (APZ) of parameter No. 1815 is set to 0 can be checked using diagnostic data Nos. 310 and 311.
After any bit of diagnostic data Nos. 310 and 311 is set to 1, it remains set to 1 until the zero point of the absolute position detector on the axis is set. Each bit of diagnostic data Nos. 310 and 311 is described below with its corresponding cause.

	#7	#6	#5	#4	#3	#2	#1	#0
310		DTH	ALP	NOF	BZ2	BZ1	PR2	PR1

PR1 One of the following parameters was changed:
No. 1820, No. 1821, No. 1850, No. 1874, No. 1875, No. 2022, No. 2084, No. 2085
PR2 The value of bit 1 (ATS) of parameter No. 8303 was changed. Alternatively, when bit 7 (SMA) of parameter No. 8302 was set to 1, APZ of the relevant axis for synchronous operation became 1.
BZ1 A battery voltage of 0 V was detected. (Inductosyn)
BZ2 A battery voltage of 0 V was detected. (Separate position detector)
NOF No offset data was output from Inductosyn.
ALP Although the α pulse coder did not indicate one turn, the zero point was set using MDI operation.
DTH An axis was released from control by using controlled axis detach signal DTCH <Gn124> or bit 7 (RMV) of parameter No. 0012.

	#7	#6	#5	#4	#3	#2	#1	#0
311		DUA	XBZ	GSG	AL4	AL3	AL2	AL1

AL1 An SV alarm (SV301 to SV305) was issued.
AL2 Disconnection alarm SV445 or SV447 was detected.
AL3 A battery voltage of 0 V was detected. (Serial pulse coder)
AL4 An abnormal speed alarm (RCAL) was detected.
GSG Disconnection alarm ignore signal NDCAL <Gn202> was changed from 1 to 0.
XBZ A battery voltage of 0 V or a count error was detected. (Separate serial position detector)
DUA When the dual position feedback function was used, the difference between the errors in the semi-closed loop and closed loop became excessive.

If a battery low alarm is issued, the cause of the issuance can be checked with diagnosis display No. 3019.

	#7	#6	#5	#4	#3	#2	#1	#0
3019			EXP	INP	ABP			

[Data type] Axis
ABP Battery low in the phase A/B
INP Battery low in the serial pulse coder (built-in position detector)
EXP Battery low in the serial separate position detector

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1815		RONx	APCx	APZx				

[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 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 pulse coder)

#6 RONx With a rotation axis, a rotary encoder for detecting an absolute position within one revolution is:
0: Not used.
1: Used.

NOTE
1 This parameter is available for only the rotary axis A type with an absolute position detector (absolute pulse coder).This function cannot be used for a rotary scale with distance-coded reference marks (serial) or for a distance coded rotary scale interface (phase A/B).
2 Set it to a rotary axis A type using a scale without rotary data.
3 Do not set it to a rotary axis A type using a scale with rotary data.
4 When this parameter is set, machine position and position on absolute position detector become uncorresponding.Consequently, the parameter APZ (No. 1815#4: indicating that the correspondence is established) is set to 0, alarm DS0300. Why the parameter APZ (No. 1815#4) is set to 0 can be checked using diagnostic data No. 310#0.

	#7	#6	#5	#4	#3	#2	#1	#0
1819							CRFx	

[Input type] Parameter input
[Data type] Bit axis

#1 CRFx When the servo alarm SV0445 (soft disconnection), SV0447 (hard disconnection (separate)), or SV0421 (dual position feedback excessive error) is issued:
0: The reference position established state is not affected.
1: The reference position unestablished state is assumed. (Bit 4 (APZ) of parameter No. 1815 is set to 0.)

Alarm and message

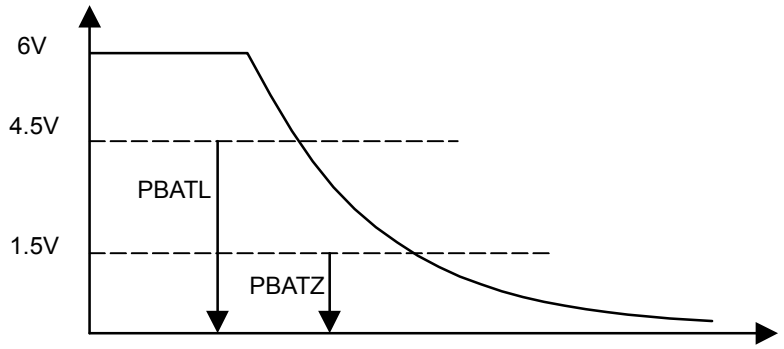
Number	Message	Description
PS0090	REFERENCE RETURN INCOMPLETE	1. The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, 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.
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 pulse coder for the first time. The battery or cable is thought to be defective. Replace the battery with the machine turned on.

Number	Message	Description
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.
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.

Caution

⚠ CAUTION

The normal voltage of the backup batteries for an absolute pulse coder is 6 V. The voltage drops as time goes by.



PBATL: Battery voltage low alarm

This alarm is output when the voltage becomes 4.5 V or less. When this alarm is output, "APC" is displayed on the status bar on the screen. After that, when the power to the CNC is turned off, then on again, the message "DS307 APC alarm: battery low 1" is displayed.

PBATZ: Battery voltage zero alarm

When the power to the CNC is turned off, then on again after the voltage becomes 1.5 V or less, the messages "DS300 APC alarm: reference position return" and "DS306 APC alarm: battery voltage 0" are displayed.

Note

NOTE

1 For an absolute position detector, batteries are used because the absolute position must be retained. When the battery voltage becomes low, a battery low alarm for the absolute position detector is displayed on the machine's operator panel or screen. If a battery voltage low alarm is displayed, replace the batteries with new ones. If the batteries are not replaced, the absolute position data in the absolute position detector is lost.

2 For the procedure for replacing batteries, refer to the method for replacing batteries in Part IV, "MAINTENANCE" in the OPERATOR'S MANUAL.

3 After replacing batteries with new ones after alarm (DS0306) (battery voltage 0) is issued, manually rotate the motor at least one turn and turn the power to the CNC and servo amplifier off, then on again. Then, set the zero point of the absolute position detector (using manual reference position return or MDI operation).
If the above operation is not performed, alarm (PS0090 or DS0309) is issued.

1.4.3 FSSB Setting

1.4.3.1 Series 0*i*-D-dedicated setting

When 0 is set in bit 0 (DFS) of parameter No. 14476, the FSSB setting is dedicated to Series 0*i*-D.
To make the setting compatible with Series 0*i*-C, set 1 in bit 0 (DFS) of parameter No. 14476. For details, see "Series 0*i*-C-compatible setting" later in this manual.

Overview

Connecting the CNC control section to servo amplifiers via a high-speed serial bus (FANUC Serial Servo Bus, or FSSB), which uses only one fiber optics cable, can significantly reduce the amount of cabling in machine tool electrical sections.
In a system using the FSSB, it is necessary to set up the following parameters to specify its axes.

- No.1023
- No.1905
- Nos.1936, 1937
- Nos. 14340 to14357, 14376 to 14391

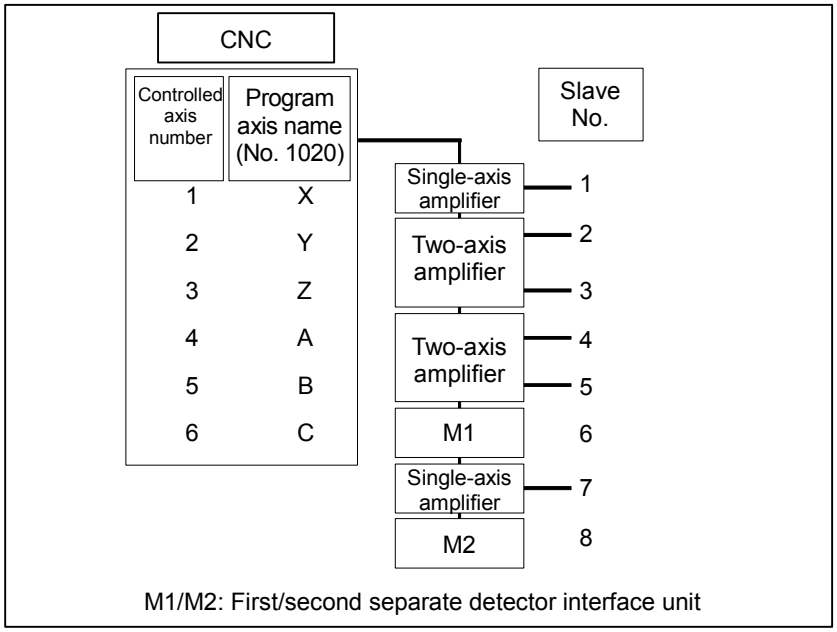
These parameters can be specified using the following methods:

1. Manual setting 1
   Parameters are defaulted according to the setting of parameter No. 1023. There is no need to specify parameter Nos. 1905, 1936 to 1937, 14340 to 14357, and 14376 to 14391. No automatic setting is used. Note that some functions are unusable.
2. Automatic setting
   Axis settings are calculated automatically according to the interrelationships between axes and amplifiers entered on the FSSB setting screen. Parameter Nos. 1023, 1905, 1936, 1937, 14340 to 14357, and 14376 to 14391 are specified automatically according to the results of the calculation.
3. Manual setting 2
   Parameter Nos. 1023, 1905, 1936, 1937, 14340 to 14357, and 14376 to 14391 are specified according to manually entered values. The user must be totally familiar with the meaning of each parameter before entering any values.

Explanation

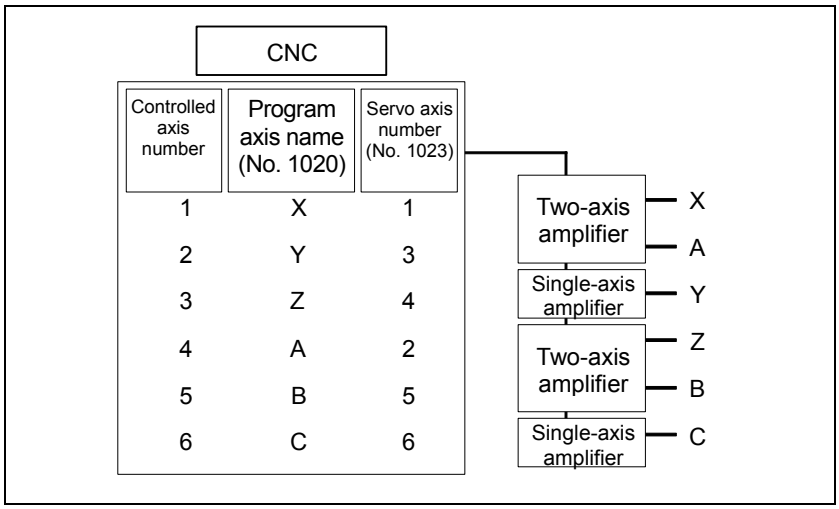
- Slave

In an FSSB-based system, a fiber optics cable is used to connect the CNC to servo amplifiers and separate detector interface units. These amplifiers and separate detector interface units are called slaves. The two-axis amplifier consists of two slaves, and the three-axis amplifier consists of three slaves. Slave numbers 1 to 10 are sequentially assigned to slaves for FSSB. Number 1 is assigned to the slave nearest to the CNC.



- Manual setting 1

The manual setting 1 is valid when the following parameter have the following values:
FMD (No. 1902#0) = 0
AES (No. 1902#1) = 0
By manual setting 1, the value set for parameter No. 1023 when the power is switched on is regarded as a slave number. Specifically, an axis for which parameter No. 1023 is set to 1 is connected to the amplifier nearest to the CNC, while an axis for which parameter No. 1023 is set to 2 is the second one from the CNC.



By manual setting 1, some of the following functions and values cannot be used, as described below. They should be used with automatic setting or manual setting 2.

- No separate detector interface unit can be used; hence, no separate position detectors can be used.
- No number can be skipped in parameter No. 1023; for example, servo axis number 3 cannot be used for any axis unless servo axis number 2 is used.
- The following servo functions cannot be used:
  - Servo HRV3 control
  - Tandem control
  - Electronic gear box (EGB) (M series)

- Automatic setting

Automatic setting can be used on the FSSB setting screen, if the following parameter is set as follows:
FMD (No. 1902#0) = 0
On the FSSB setting screen, automatic setting should be enabled by means of the following procedure:

1 On the amplifier setting screen, information on servo amplifiers and pulse modules is displayed in ascending order of slave number.
2 Specify the number of the controlled axis to be connected for each slave. At this time, the corresponding controlled axis name is displayed at the right of the number. (For other than pulse modules)
3 Select the axis setting screen and specify function data such as connector numbers for the pulse module for each controlled axis.
4 Press the [SETTING] soft key to perform automatic setting. If setting data contains an error, a warning message is displayed. Set correct data again.

In this way, parameter Nos. 1023, 1905, 1936, 1937, 14340 to 14357, and 14376 to 14391 are set according to the results of automatic calculation. In addition, ASE (No. 1902#1) is set to 1 to indicate that each parameter has been set up. Switching the power off then back on again causes axis setting to be performed according to these parameter settings.
For details of the FSSB setting screen, see the FSSB data display and setting procedure, described below.

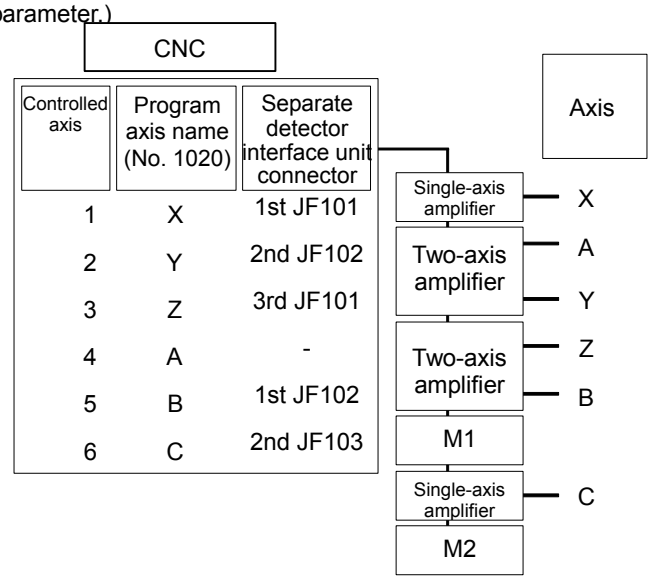
- Manual setting 2

After parameter FMD (No. 1902#0) is set to 1 or automatic setting has been terminated (ASE (No. 1902#1) is set to 1), manual setting 2 for each parameter for axis setting can be performed.
To perform manual setting 2, set parameter Nos. 1023, 1905, 1936, 1937, 14340 to 14357, and 14376 to 14391. Refer to the Parameter Manual for the definition of each parameter.

Example of setting parameters

- When separate detector interface units are connected

Example of axis configuration and manual setting 2
(Refer to the corresponding description for the meaning of each parameter.)



M1/M2: First/second separate detector interface unit

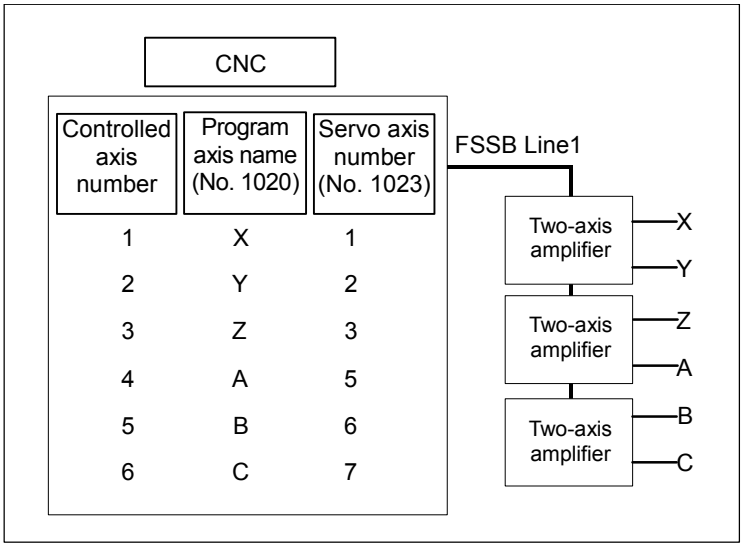
No.	1902#0 FMD
	1

No.	1023	1905#6 PM1	1905#7 PM2	1936	1937
X	1	1	0	0	-
Y	3	0	1	-	1
Z	4	0	1	-	0
A	2	0	0	-	-
B	5	1	0	1	-
C	6	0	1	-	2

No.	14340	14341	14342	14343	14344	14345	14346	14347
	0	1	2	3	4	64	5	-56
No.	14348 to 14357							
	-96							
No.	14376	14377	14378	14379	14380	14381	14382	14383
	0	4	32	32	32	32	32	32
No.	14384	14385	14386	14387	14388	14389	14390	14391
	3	2	5	32	32	32	32	32

- For servo HRV3 control

When servo HRV3 control is used with the servo software 90E5 series, specify numbers other than a multiple of 4 in parameter No. 1023 as servo axis numbers.



No.	1902#0 FMD
	1

No.	14340	14341	14342	14343	14344	14345
	0	1	2	4	5	6
No.	14346 to 14357					
	-96					

NOTE

When servo HRV3 control is used, the number of units connected to FSSB is limited. For details, refer to “Connection to the Servo Amplifiers” in “Connection Manual (Hardware)” (B-64303EN).

FSSB display and setting procedure

- Display

The FSSB setting screen displays FSSB-based amplifier and axis information. This information can also be specified by the operator.

1 Press the SYSTEM function key.

2 To display [FSSB], press the next menu page key several times.

3 Pressing the [FSSB] soft key causes the [AMPLIFIER SETTING] screen (or the previously selected FSSB setting screen) to appear, with the following soft keys displayed.



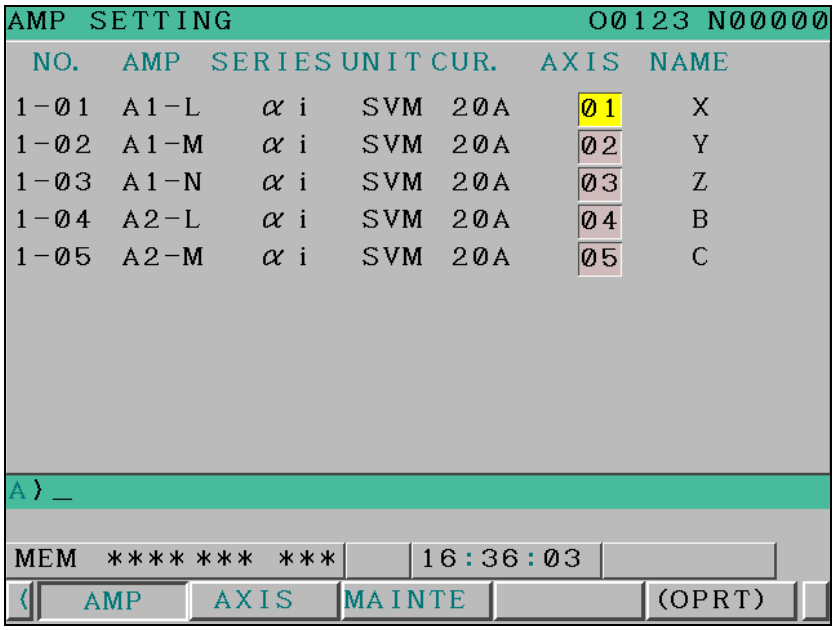
The FSSB setting screens include: [AMPLIFIER SETTING], [AXIS SETTING], and [AMPLIFIER MAINTENANCE]

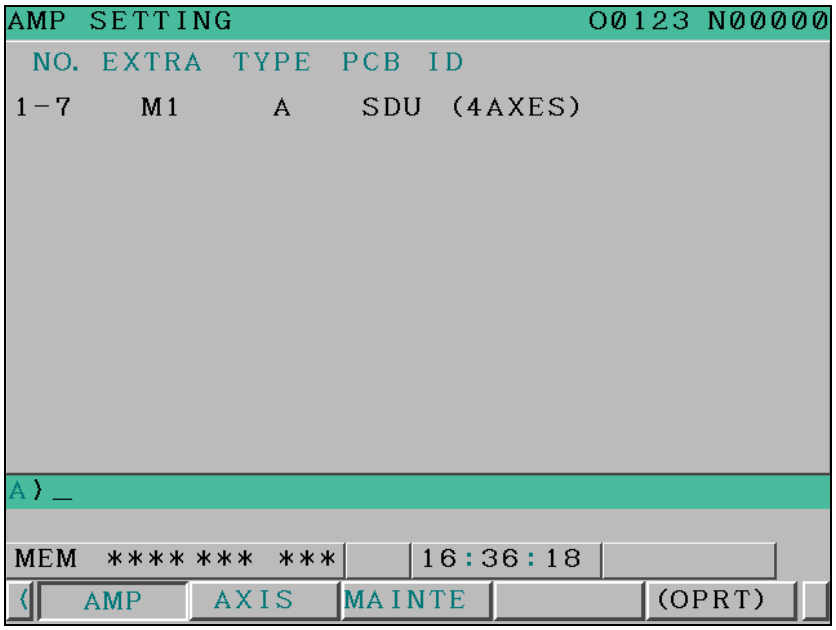
Pressing the [AMP] soft key causes the [AMPLIFIER SETTING] screen to appear.
Pressing the [AXIS] soft key causes the [AXIS SETTING] screen to appear.
Pressing the [MAINTE] soft key causes the [AMPLIFIER MAINTENANCE] screen to appear.

(1) Amplifier setting screen

The amplifier setting screen consists of two sections: the first section displays information about the amplifiers, while the second section displays information about the separate detector interface units.

Use page keys PAGE and PAGE to switch between the two sections.



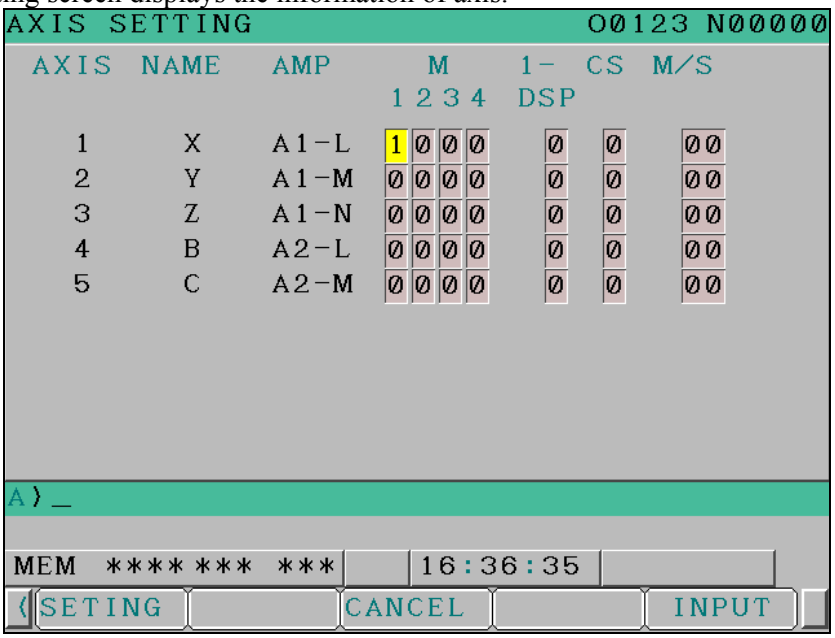


The amplifier setting screen consists of the following items:

- NO. : Slave number
  For FSSB, the serial numbers assigned to up to 10 slaves (up to 8 amplifiers and up to 2 separate detector interface units) connected by the FSSB are displayed. Number 1 is assigned to the slave nearest to the CNC.
- AMP : Amplifier type
  The amplifier type display consists of the letter A, which stands for "amplifier", a number that indicates the placing of the amplifier, as counted from that nearest to the CNC, and a letter such as L (first axis), M (second axis), or N (third axis) indicating the placing of the axis in the amplifier.
- AXIS : Controlled axis number
  The number of the controlled axis specified in each of parameters Nos. 14340 to 14357 is displayed.
  If a number specified in these parameters falls outside the range of 1 and the maximum number of controlled axes, 0 is displayed.
- NAME : Controlled axis name
  The axis name assigned to a parameter (No. 1020) corresponding to a particular controlled axis number is displayed.
  When the extended axis name function is enabled, the axis name characters specified for parameters Nos. 1025 and 1026 are also displayed.
  If the controlled axis number is 0, - is displayed.
- The following items are displayed as amplifier information:
  - UNIT : Servo amplifier unit type
  - SERIES : Servo amplifier series
  - CUR. : Maximum rating current
- The following items are displayed as separate detector interface unit information:
  - EXTRA
    This consists of the letter M, which stands for "separate detector interface unit", and a number indicating the placing of the separate detector interface unit, as counted from that nearest to the CNC.
  - TYPE
    This is a letter indicating the type of the separate detector interface unit.

- PCB ID
  This consists of four digits indicating the separate detector interface unit ID (hexadecimal). The separate detector interface unit ID is followed by SDU (8AXES) when 8-axes separate detector interface unit or SDU (4AXES) when 4-axes separate detector interface unit.

(2) Axis setting screen
The axis setting screen displays the information of axis.

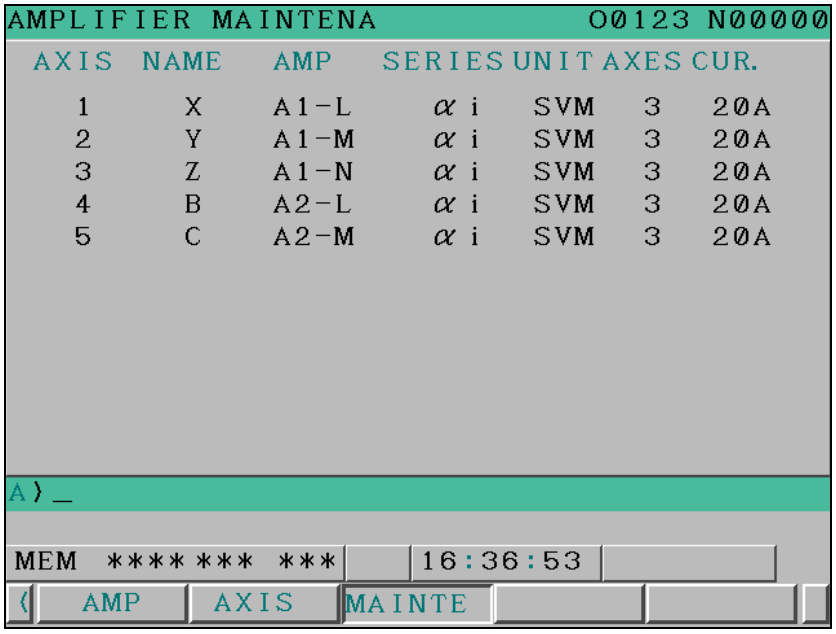


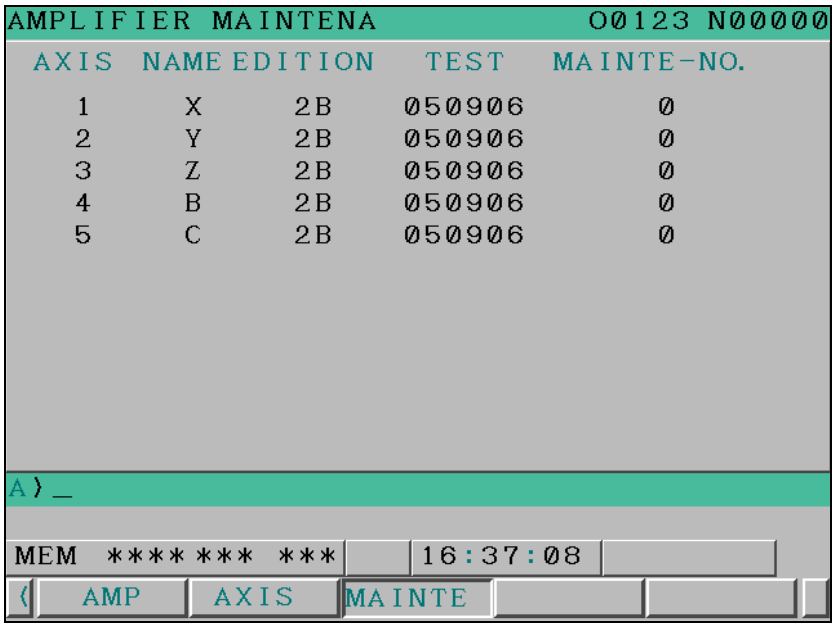
The axis setting screen displays the following items:

- AXIS : Controlled axis number
  This item is the placing of the NC controlled axis.
- NAME : Controlled axis name
- AMP : Type of the amplifier connected to each axis
- M1:Connector number for separate detector interface unit 1
  The connector number for separate detector interface unit 1 that is retained in the SRAM is displayed.
- M2:Connector number for separate detector interface unit 2
  The connector number for separate detector interface unit 2 that is retained in the SRAM is displayed.
- 1-DSP
  When the number of servo HRV3 controlled axes controlled by one DSP is limited, the number of axes that can be controlled by one DSP is displayed. The number is retained in the SRAM. When the number of axes is not limited, "0" is displayed.
- CS : Cs contour controlled axis
  The value retained in the SRAM is displayed. A spindle number is displayed for the Cs contour controlled axis.
- TNDM
  The value retained in the SRAM is displayed. Consecutive odd and even numbers for the master and slave axes for tandem control are displayed. This item is not used in Series 0*i* Mate-D.

(3) Amplifier maintenance screen

The amplifier maintenance screen displays maintenance information for servo amplifiers. This screen consists of the following two pages, either of which can be selected by pressing the PAGE or PAGE.





The amplifier maintenance screen displays the following items:

- AXIS : Controlled axis number
- NAME : Controlled axis name
- AMP : Type of amplifier connected to each axis
- SERIES : Servo amplifier series of an amplifier connected to each axis
- UNIT : Unit type of a servo amplifier connected to each axis
- AXES : Maximum number of axes controlled by an amplifier connected to each axis
- CUR. : Maximum rating current for amplifiers connected to each axis
- EDITION : Unit version number of an amplifier connected to each axis

- TEST : Date of test performed on an amplifier connected to each axis
  Example) 030123 = January 23, 2003
- MAINTE-No. : Engineering change number for an amplifier connected to each axis

- Setting

On an FSSB setting screen (other than the amplifier maintenance screen), pressing the [(OPRT)] soft key displays the following soft keys:



To enter data, place the machine in MDI mode or the emergency stop state, position the cursor to the point where a desired item is to be input, then enter the desired data and press the [INPUT] soft key (or the 

key on the MDI panel).

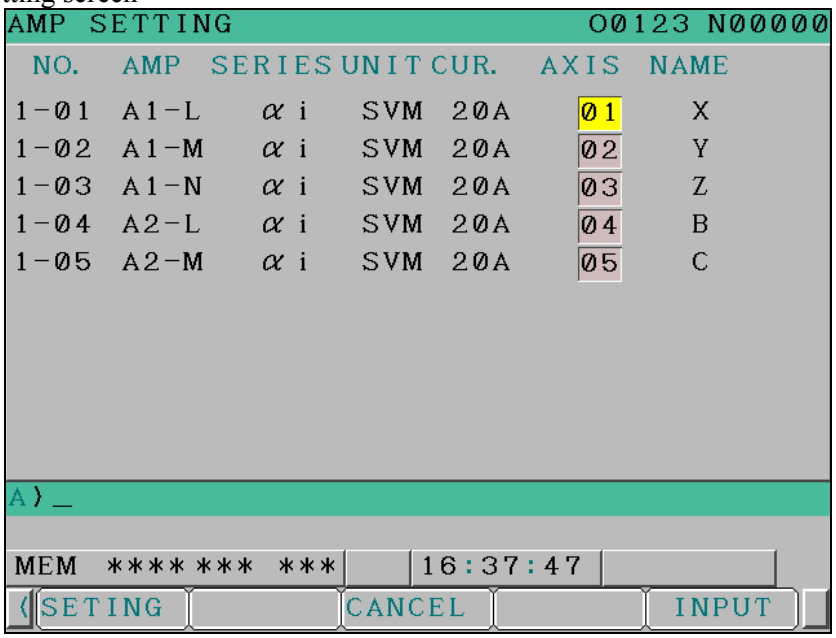
When the [SETTING] key is pressed after data has been entered, a warning message is displayed if the entered data contains an error. When the data is valid, the corresponding parameter (Nos.1023, 1905, 1936, 1937, 14340 to 14357, and 14376 to 14391) is set up.

To restore the previous value of a parameter if, for example, an entered value is incorrect, press the [CANCEL] soft key.

When the power is switched on, values are read from the parameters and displayed on the screen.

> **⚠ CAUTION**
> 1 For the parameters to be specified on the FSSB setting screen, do not attempt to enter values on the parameter screen using the MDI or a G10 command. Use only the FSSB screen to enter values for these parameters.
> 2 If pressing the [SETTING] soft key causes a warning, retry data entry, or press the [CANCEL] key to clear the warning message. Note that pressing the reset key does not clear the warning message.

(1) Amplifier setting screen

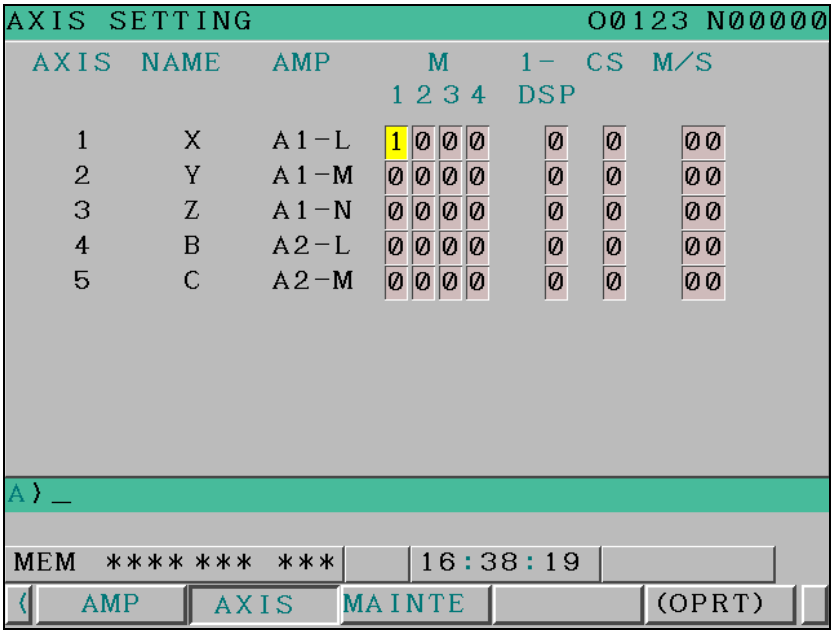


The amplifier setting screen displays the following items:

- AXIS : Controlled axis number
  For this item, enter a value of between 1 and the maximum number of controlled axes. If a number that falls outside this range is entered, the warning message FORMAT ERROR appears.

If the entered controlled axis number is duplicate or 0, the warning message DATA IS OUT OF RANGE appears when the [SETTING] soft key is pressed to assert the entered value. In this case, no value can be entered for the parameter.

(2) Axis setting screen



On the axis setting screen, the following items can be specified:

- M1:Connector number for separate detector interface unit 1
- M2:Connector number for separate detector interface unit 2
  For an axis that uses each separate detector interface unit, enter a connector number using a number 1 to 8 (maximum number of connectors on a separate detector interface unit).
  When a separate detector interface unit is not used, enter "0".
  When a separate detector interface unit is not connected and a number outside the valid range is entered, the warning message ILLEGAL DATA is displayed. When a separate detector interface unit is connected and a number outside the valid range is entered, the warning message DATA IS OUT OF RANGE is displayed.
- 1-DSP
  Enter the number of servo HRV3 controlled axes that can be controlled by one DSP when that number is limited.
  For servo HRV3 controlled axes Setting: 3
  The same value is set for axes other than Cs contour controlled axes.
  If a number other than 0 or 3 is entered, the message FORMAT ERROR is displayed.
- CS : Cs contour controlled axis
  Enter spindle number (1 to 3) for the Cs contour controlled axis. If a number other than 0 to 3 is entered, the message DATA IS OUT OF RANGE is displayed.
- TNDM
  Enter odd and even numbers for axes for tandem control or EGB. These numbers must be consecutive and within a range between 1 and the number of controlled axes. If a number that falls outside the valid range is entered, the message DATA IS OUT OF RANGE is displayed.
  This item is not used in Series 0*i* Mate-D.

When the [SETTING] soft key is pressed on the axis setting screen after data entry, the message DATA IS OUT OF RANGE is displayed if any of the following conditions is satisfied.

- Controlled axes cannot be connected to amplifiers and separate detector interface units according to the settings.
- A value other than 0 is specified for both M1 and M2 for an axis.
- A value other than 0 is specified for both CS and TNDM for an axis.

- 1-DSP is 3 and TNDM is a multiple of 4 for an axis.
- A duplicate value is specified for M1.
- A duplicate value is specified for M2.
- A duplicate value is specified for CS.
- A duplicate value is specified for TNDM.
- An invalid master/slave axis pair is specified for TNDM.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
14476								DFS

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 DFS The FSSB enters:
0: The FS0*i*-D-specific mode.
1: The FS0*i*-C-compatible mode.

	#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 bit 0 (DFS) of parameter No. 14476 is 0:
If the relationship between the axis and the amplifier and the like are defined on the FSSB setting screen, parameters No. 1023, No. 1905, Nos. 1936 and 1937, Nos. 14340 to 14357, and Nos. 14376 to 14391 are automatically set.)
(When bit 0 (DFS) of parameter No. 14476 is 1:
If the relationship between the axis and the amplifier and the like are defined on the FSSB setting screen, parameters No. 1023, No. 1905, Nos. 1910 to 1919, and Nos. 1936 and 1937 are automatically set.)
1: Manual setting 2 mode.
(When bit 0 (DFS) of parameter No. 14476 is 0:
Manually set parameters No.1023, No.1905, Nos.1936 and 1937, Nos.14340 to 14357, and Nos.14376 to 14391.)
(When bit 0 (DFS) of parameter No. 14476 is 1:
Manually set parameters No.1023, No.1905, Nos.1910 to 1919, and Nos. 1936 and 1937.)

#1 ASE When automatic setting mode is selected for FSSB setting (when the FMD parameter (bit 0 of 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.

	#7	#6	#5	#4	#3	#2	#1	#0
1905	PM2	PM1						

[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..

#6 PM1 The first separate detector interface unit is:
0: Not used.
1: Used.

#7 PM2 The second separate detector interface unit is:
0: Not used.
1: Used.

NOTE
When automatic setting mode is selected for FSSB setting (when the parameter FMD (No.1902#0) is set to 0), this parameter is automatically set when input is performed with the FSSB setting screen. When manual setting 2 mode is selected for FSSB setting (when the parameter FMD (No.1902#0) is set to 1), this parameter must be set directly. When a separate detector interface unit is used, a connector number must be set in the corresponding parameter (No.1936 or No.1937).

1936	Connector number of the first separate detector interface unit
1937	Connector number of the second 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 7
This parameter sets the connector number corresponding to the connector connected when using the separator detector interface unit set by bits 6 (PM1) and 7 (PM2) of parameter No. 1905 minus 1. That is, set 0 to 7 for connector numbers 1 to 8, respectively. Set 0 for the axis for which the separator detector interface unit is not used.
Use successive numbers for one separator detector interface unit. Do not omit a intermediate number.

Example)

Controlled axis	Connector number for the first separate detector interface unit	Connector number for the second separate detector interface unit	No.1936	No.1937	PM2x, PM1x (No.1905#7, #6)
X	1	Not used	0	0	0, 1
Y	Not used	2	0	1	1, 0
Z	Not used	1	0	0	1, 0
A	Not used	Not used	0	0	0, 0

NOTE
When automatic setting mode is selected for FSSB setting (when the parameter FMD (No.1902#0) is set to 0), these parameters are automatically set when input is performed with the FSSB setting screen. When manual setting 2 mode is selected for FSSB setting (when the parameter FMD (No.1902#0) is set to 1), these parameters must be set directly.

14340	**ATR value corresponding to slave 01 on FSSB**
to	
14357	**ATR value corresponding to slave 18 on FSSB**

NOTE
When these parameters are set, the power must be turned off before operation is continued..

[Input type] Parameter input
[Data type] Byte
[Valid data range] 0 to 7, 64, -56, -96

Each of these parameters sets the value (ATR value) of the address translation table corresponding to each of slave 1 to slave 18 on FSSB.
The slave is a generic term for servo amplifiers and separate detector interface units connected via an FSSB optical cable to the CNC. Numbers 1 to 18 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, or nonexistent.

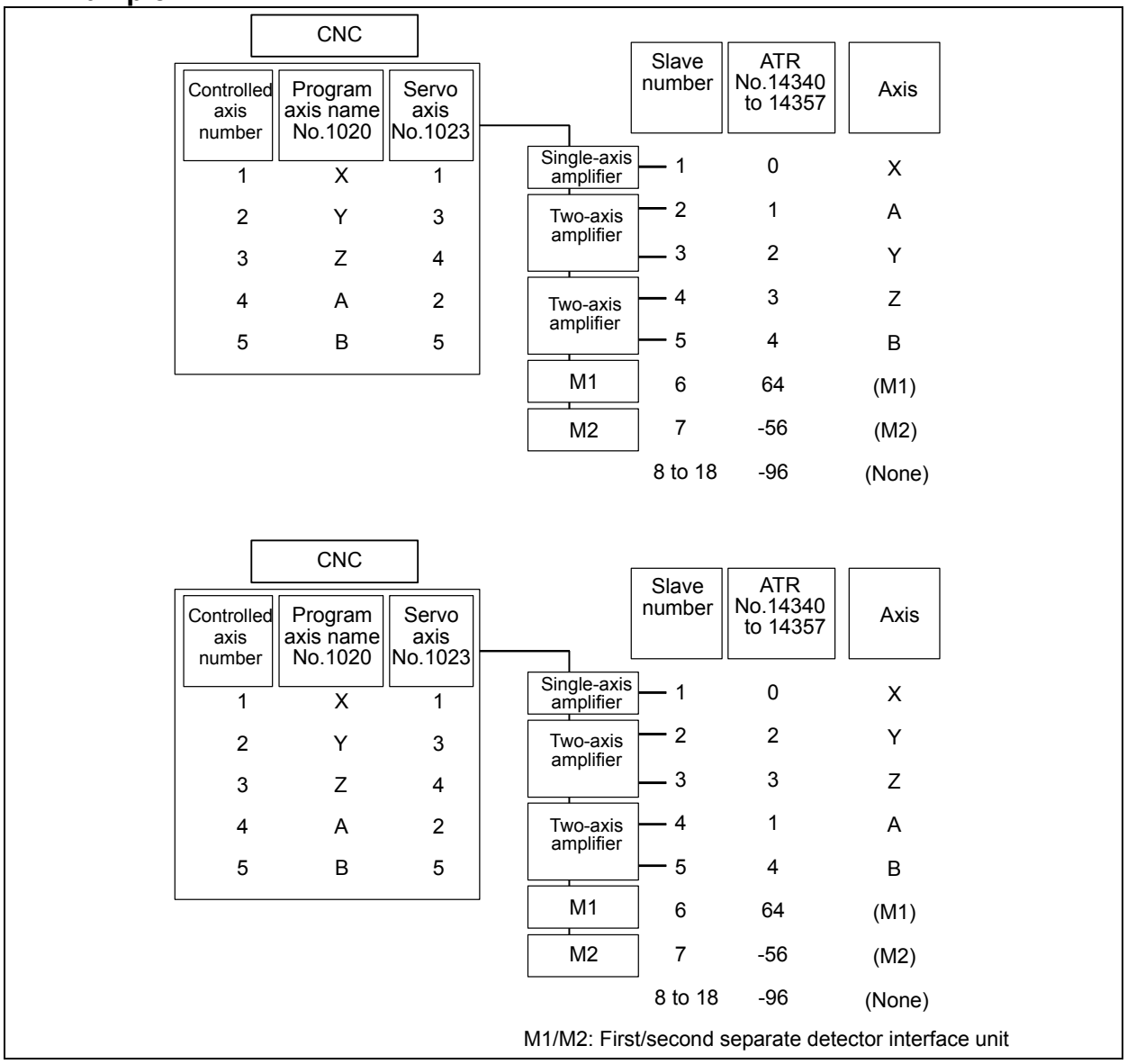
- When the slave is an amplifier:
  Set a value obtained by subtracting 1 from the setting of parameter No. 1023 for the axis to which the amplifier is assigned.
- When the slave is a separate detector interface unit:
  Set 64 for the first separate detector interface unit (connected near the CNC), and set -56 for the second separate detector interface unit (connected far from the CNC).
- When the slave is nonexistent:
  Set -96.

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. That is, as the address conversion table value for a nonexistent slave, set the value obtained by subtracting 1 from the setting of parameter No. 1023 for the EGB dummy axis, instead of -96.

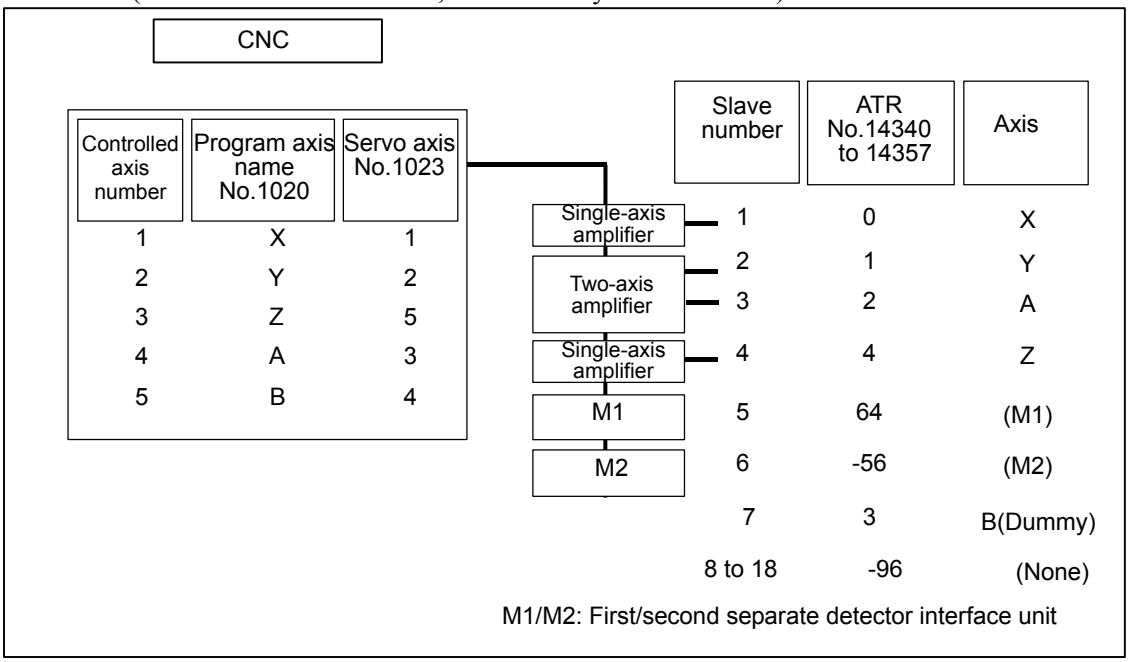
2 When the FSSB is set to the automatic setting mode (when the parameter FMD (No.1902#0) is set to 0), parameter Nos. 14340 to 14357 are automatically set as data is input on the FSSB setting screen. When the manual setting 2 mode is set (when the parameter FMD (No.1902#0) is set to 1), be sure to directly set values in parameter Nos. 14340 to 14357.

- **Example of axis configuration and parameter settings**
  Example 1



Example 2

Example of axis configuration and parameter settings when the electronic gear box (EGB) function (M series) is used
(EGB slave axis: A-axis, EGB dummy axis: B-axis)



14376	**ATR value corresponding to connector 1 on the first separate detector interface unit**
to	**to**
14383	**ATR value corresponding to connector 8 on the first separate detector interface unit**
14384	**ATR value corresponding to connector 1 on the second separate detector interface unit**
to	**to**
14391	**ATR value corresponding to connector 8 on the third 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
[Valid data range] 0 to 7, 32

Each of these parameters sets the value (ATR value) of the address translation table corresponding to each connector on a separate detector interface unit.
In each of these parameters, set a value obtained by subtracting 1 from the setting of parameter No. 1023 for the axis connected to a connector on a separate detector interface unit.
When there are axes for which settings are made to use a separate detector interface unit (bit 6 (PM1x) of parameter No. 1905 is set to 1 or bit 7 (PM2x) of parameter No. 1905 is set to 1, set 32 for connectors not used.

NOTE
1 Set this parameter when using a separate detector interface unit.

NOTE
2 When the FSSB is set to the automatic setting mode (when the parameter FMD (No.1902#0) is set to 0), parameter Nos. 14376 to 14391 are automatically set as data is input on the FSSB setting screen. When the manual setting 2 mode is set (when the parameter FMD (No.1902#0) is set to 1), be sure to directly set values in parameter Nos. 14376 to 14407.

Alarm and message

Number	Message	Description
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. Or, the FSSB connection cable was disconnected or broken. The amplifier was turned off . 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. Or, the connection command for an amplifier is incorrect. The parameter 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.
SV1067	FSSB:CONFIGURATION ERROR(SOFT)	An FSSB configuration error occurred (detected by software). The connected amplifier type is incompatible with the FSSB setting value.
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	The axis setting has not completed yet, in an automatic setting mode.
SV5197	FSSB:OPEN TIME OUT	Servo initialization has not completed successfully. It is probable that an optical cable failed or a connection between the amplifier and another module failed.
SV5311	FSSB:ILLEGAL CONNECTION	1. This alarm is issued if axes, whose servo axis numbers (parameter No. 1023) are even and odd numbers, are allocated to the amplifiers connected to the FSSBs of different paths. 2. This alarm is issued if an attempt is made to set up for use of the pulse modules connected to the FSSBs of different paths. And the system did not satisfy the requirements for performing HIGH SPEED HRV control.

1.4.3.2 Series 0*i*-C-compatible setting

When 1 is set in bit 0 (DFS) of parameter No. 14476, the FSSB setting is compatible with Series 0*i*-C.

Overview

This subsection describes how to set the parameters needed for Series 0*i*-D to drive the βiSVSP and βiSV2 axes that support only the FS0*i*-C compatible mode.

To drive the βiSVSP and βiSV2 axes that support only the FS0*i*-C compatible mode, set the parameters as instructed below, after setting 1 in bit 0 of parameter No. 14476 and turning the power off and then back on.

Connecting the CNC control section to servo amplifiers via a high–speed serial bus (FANUC Serial Servo Bus, or FSSB), which uses only one fiber optics cable, can significantly reduce the amount of cabling in machine tool electrical sections.

In a system using the FSSB, it is necessary to set up the following parameters to specify its axes. (The other parameters should be specified as usual.)

- No.1023
- No.1905
- Nos.1910 to 1919
- Nos.1936 and 1937

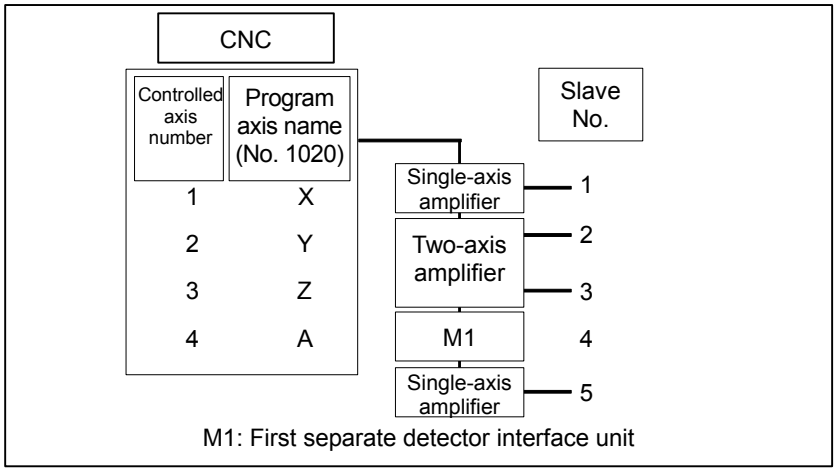
These parameters can be specified using the following methods:

1. Manual setting 1
   Parameters are defaulted according to the setting of parameter No. 1023. There is no need to specify parameter Nos. 1905, 1910 to 1919, 1936 and 1937. No automatic setting is used. Note that some functions are unusable.
2. Automatic setting
   Axis settings are calculated automatically according to the interrelationships between axes and amplifiers entered on the FSSB setting screen. Parameter Nos. 1023, 1905, 1910 to 1919, 1936, and 1937 are specified automatically according to the results of the calculation.
3. Manual setting 2
   Parameter Nos. 1023, 1905, 1910 to 1919, 1936, and 1937 are specified according to manually entered values. The user must be totally familiar with the meaning of each parameter before entering any values.

Explanation

- Slave

In an FSSB–based system, a fiber optics cable is used to connect the CNC to servo amplifiers and separate detector interface units. These amplifiers and separate detector interface units are called slaves. The two–axis amplifier consists of two slaves, and the three–axis amplifier consists of three slaves. The slaves are numbered 1, 2, ..., 10 (slave number) sequentially, with that nearest to the CNC starting at number 1.

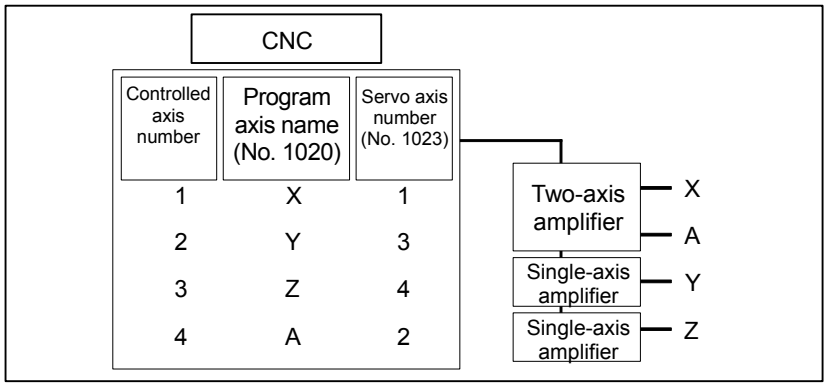


- Manual setting 1

The manual setting 1 is valid when the following parameter have the following values:

FMD(No.1902#0)="0"
ASE(No.1902#1)="0"
Parameter Nos. 1910 to 1919 = all 0s

By manual setting 1, the value set for parameter No. 1023 when the power is switched on is regarded as a slave number. Specifically, an axis for which parameter No. 1023 is set to 1 is connected to the amplifier nearest to the CNC, while an axis for which parameter No. 1023 is set to 2 is the second one from the CNC.



By manual setting 1, some of the following functions and values cannot be used, as described below. They should be used with automatic setting or manual setting 2.

- No separate detector interface unit can be used; hence, no separate position detectors can be used.
- No number can be skipped in parameter No. 1023; for example, number 3 cannot be used for any axis unless number 2 is used.
- The following servo functions cannot be used:
  - High–speed current loop
  - High–speed interface axis

- Automatic setting

Automatic setting can be used on the FSSB setting screen, if the following parameter is set as follows:

FMD(No.1902#0)="0"

On the FSSB setting screen, automatic setting should be enabled by means of the following procedure:

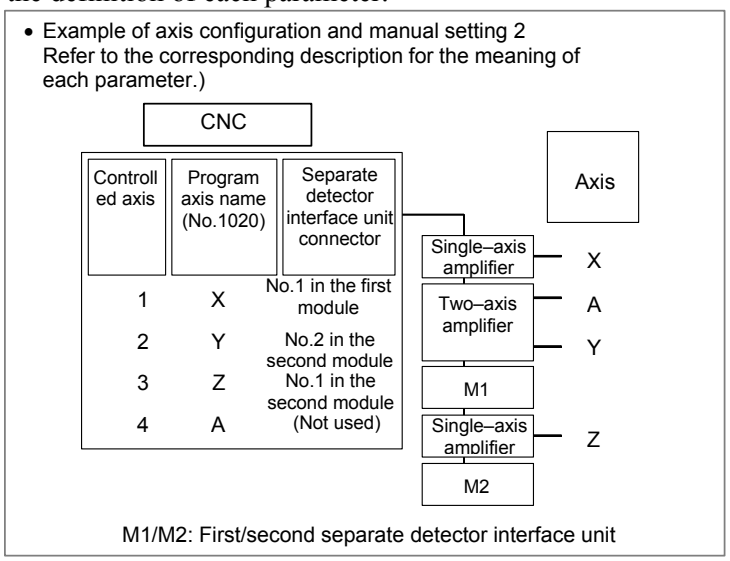
1. On the amplifier setting screen, specify the axis number of a controlled axis to be connected to each amplifier.
2. Press the [SETING] soft key. (If a warning message is displayed, restart from step 1.)
3. On the axis setting screen, specify information about each axis, such as a separate detector interface unit connector No.

4. Press the [SETING] soft key. (If a warning message is displayed, repeat the procedure, starting from step 3.)

In this way, parameter Nos. 1023, 1905, 1910 to 1919, 1936, and 1937 are set according to the results of automatic calculation. In addition, bit 1 (ASE) of parameter No. 1902 is set to 1 to indicate that each parameter has been set up. Switching the power off then back on again causes axis setting to be performed according to these parameter settings.
For details of the FSSB setting screen, see the FSSB data display and setting procedure, described below.

- Manual setting 2

If the following parameter is set, manual setting 2 can be used for each parameter axis setting.
FMD(No.1902#0)="0"
To perform manual setting 2, set parameter Nos. 1023, 1905, 1910 to 1919, 1936, and 1937. Refer to the Parameter Manual for the definition of each parameter.



No.	1902#0 FMD
	1

No.	1910	1911	1912	1913	1914	1915	1916	1917	1918	1919
	0	1	2	16	3	48	40	40	40	40

No.	1023	1905#0 FSL	1905#6 PM1	1905#7 PM2	1936	1937
X	1	0	1	0	0	0
Y	3	0	0	1	0	1
Z	4	1	0	1	0	0
A	2	1	0	0	0	0

FSSB display and setting procedure

- Display

The FSSB setting screen displays FSSB–based amplifier and axis information. This information can also be specified by the operator.

1 Press the SYSTEM function key.

2 To display [FSSB], press the continuous menu key ▷ several times.

3 Pressing the [FSSB] soft key causes the [AMPLIFIER SETTING] screen (or the previously selected FSSB setting screen) to appear, with the following soft keys displayed.

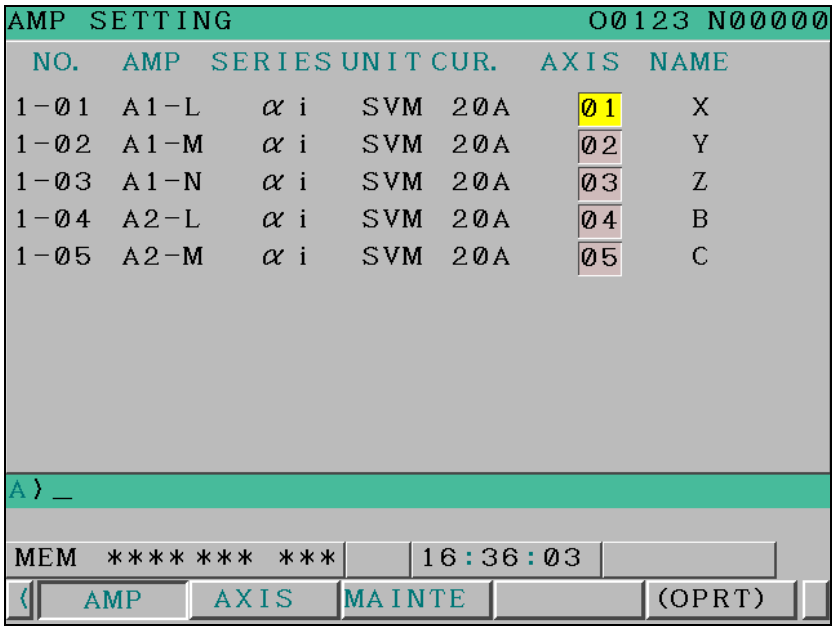


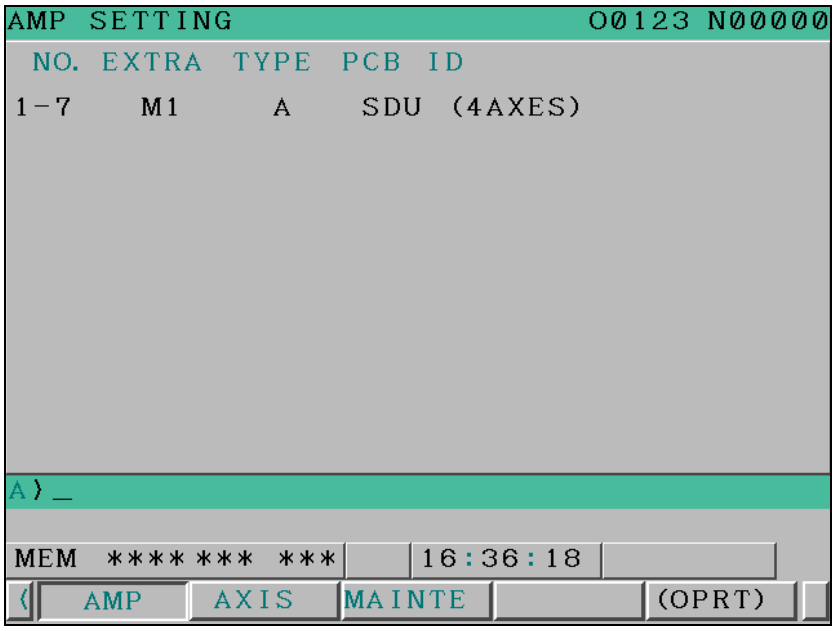
The FSSB setting screens include: [AMPLIFIER SETTING], [AXIS SETTING], and [AMPLIFIER MAINTENANCE]

Pressing the [AMP] soft key causes the [AMPLIFIER SETTING] screen to appear.
Pressing the [AXIS] soft key causes the [AXIS SETTING] screen to appear.
Pressing the [MAINTE] soft key causes the [AMPLIFIER MAINTENANCE] screen to appear.

(1) Amplifier setting screen

The amplifier setting screen consists of two sections: the first section displays information about the amplifiers, while the second section displays information about the separate detector interface units.



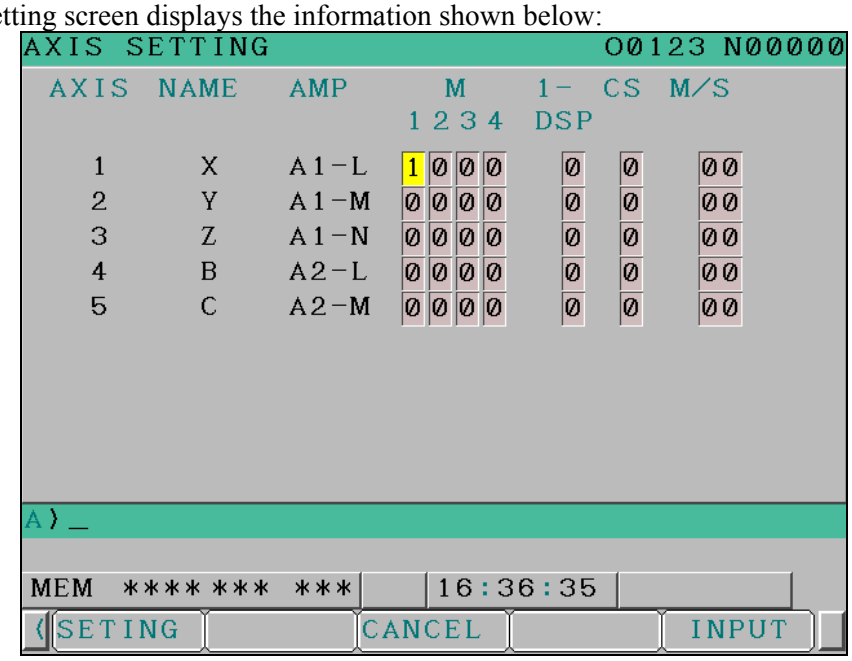


The amplifier setting screen consists of the following items:

- NO.: Slave number
  The numbers of up to ten slaves (up to six amplifiers and up to two separate detector interface units) connected via the FSSB are displayed sequentially, with the one nearest the CNC being number 1.
- AMP: Amplifier type
  The amplifier type display consists of the letter A, which stands for “amplifier”, a number that indicates the placing of the amplifier, as counted from that nearest to the CNC, and a letter such as L (first axis) or M (second axis) indicating the placing of the axis in the amplifier.
- AXIS: Controlled axis number
  Set the controlled axis numbers for slaves 1 to 10.
  If a number specified in these parameters falls outside the range of 1 and the maximum number of controlled axes, 0 is displayed.
- NAME: Controlled axis name
  The axis name assigned to a parameter (No. 1020) corresponding to a particular controlled axis number is displayed. If the controlled axis number is 0, – is displayed.
- The following items are displayed as amplifier information:
  - UNIT: Servo amplifier unit type
  - SERIES: Servo amplifier series
  - CUR.: Maximum rating current
- The following items are displayed as separate detector interface unit information:
  - EXTRA
    This consists of the letter M, which stands for “separate detector interface unit”, and a number indicating the placing of the separate detector interface unit, as counted from that nearest to the CNC.
  - TYPE
    This is a letter indicating the type of the separate detector interface unit.
  - PCB ID
    This consists of four digits indicating the separate detector interface unit ID (hexadecimal). The separate detector interface unit ID is followed by DETECTOR (8AXES) when 8–axes separate detector interface unit or DETECTOR (4AXES) when 4–axes separate detector interface unit.

(2) Axis setting screen

The axis setting screen displays the information shown below:

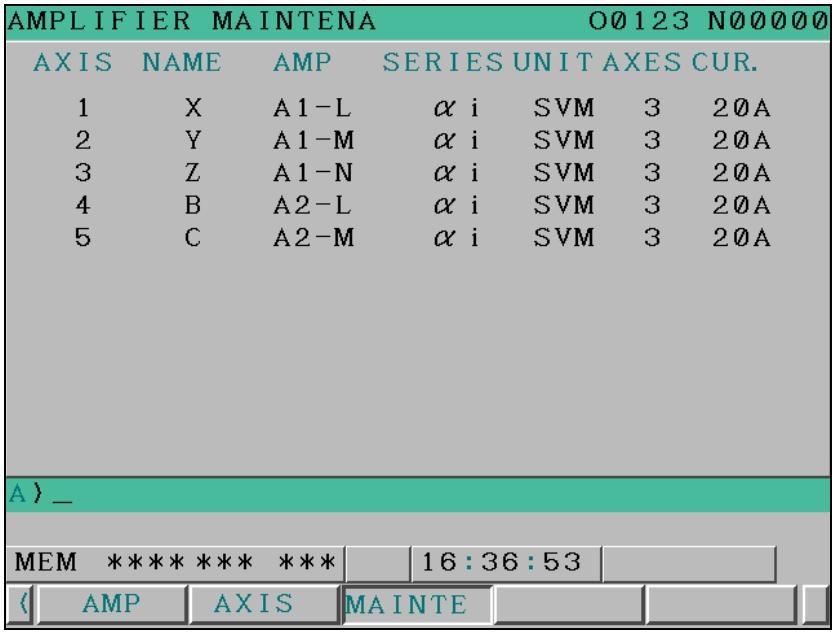


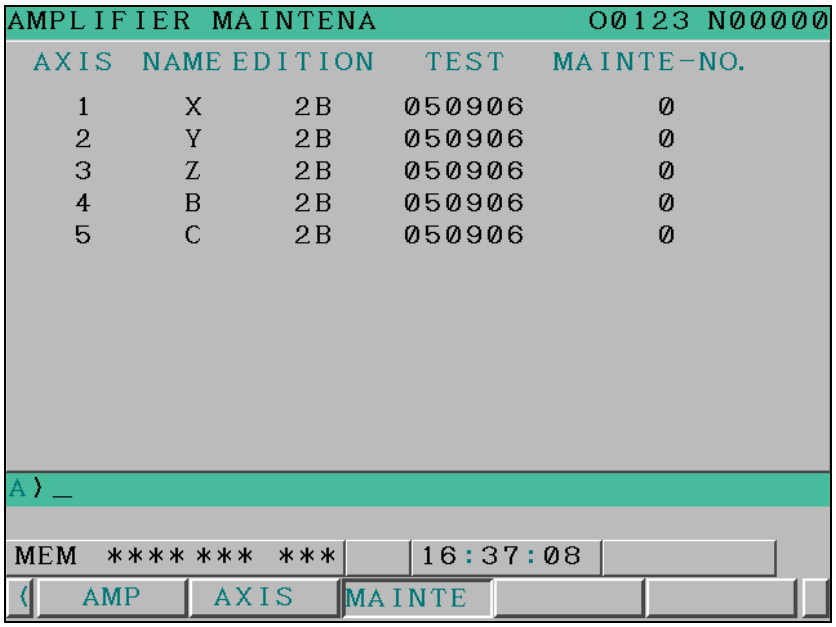
The axis setting screen displays the following items:

- AXIS: Controlled axis number
  This item is the placing of the NC controlled axis.
- NAME: Controlled axis name
- AMP: Type of the amplifier connected to each axis
- M1: Connector number for separate detector interface unit
  When using separate detector interface unit 1, set the connector number of separate detector interface unit 1 for each axis.
- M2: Connector number for separate detector interface unit
  When using separate detector interface unit 2, set the connector number of separate detector interface unit 2 for each axis.
- 1–DSP
  This item is the value specified in bit 0 (DSP) of parameter No. 1904. It is 1 for an axis (such as a learning control axis, high–speed current loop axis, or high–speed interface axis) that exclusively uses a DSP, which is usually shared by two axes.
- CS: Cs contour controlled axis
  To exert Cs contour control, set 1 for the target axis.
- TNDM
  To exert tandem control, set consecutive odd and even numbers for the master and slave axes.
  This item is not used in Series 0*i* Mate-D.

(3) Amplifier maintenance screen

The amplifier maintenance screen displays maintenance information for servo amplifiers. This screen consists of the following two pages, either of which can be selected by pressing the PAGE, PAGE key.





The amplifier maintenance screen displays the following items:

- AXIS: Controlled axis number
- NAME: Controlled axis name
- AMP: Type of amplifier connected to each axis
- SERIES: Servo amplifier series of an amplifier connected to each axis
- UNIT: Unit type of a servo amplifier connected to each axis
- AXES: Maximum number of axes controlled by an amplifier connected to each axis
- CUR.: Maximum rating current for amplifiers connected to each axis
- EDITION: Unit version number of an amplifier connected to each axis
- TEST: date of test performed on an amplifier connected to each axis
  Example) 010123 = January 23, 2001
- MAINTE–No.: engineering change number for an amplifier connected to each axis

- Setting

On an FSSB setting screen (other than the amplifier maintenance screen), pressing the [(OPRT)] soft key displays the following soft keys:



To enter data, place the machine in MDI mode or the emergency stop state, position the cursor to the point where a desired item is to be input, then enter the desired data and press the [INPUT] soft key (or the 

key on the MDI panel).

When the [SETING] key is pressed after data has been entered, a warning message is displayed if the entered data contains an error. When the data is valid, the corresponding parameter is set up.

To restore the previous value of a parameter if, for example, an entered value is incorrect, press the [CANCEL] soft key.

When the power is switched on, values are read from the parameters and displayed on the screen.

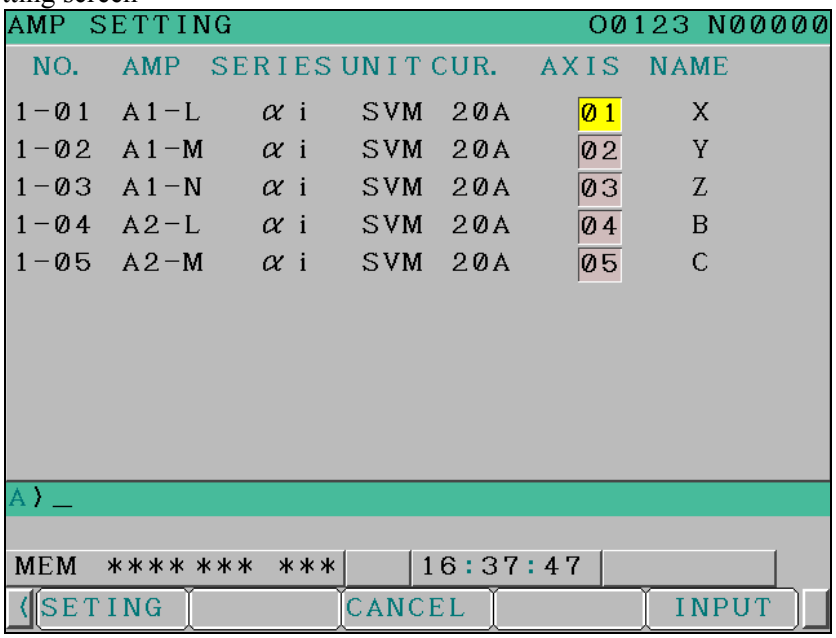
⚠ CAUTION

1 For the parameters to be specified on the FSSB setting screen, do not attempt to enter values on the parameter screen using the MDI or a G10 command. Use only the FSSB screen to enter values for these parameters.

⚠ CAUTION

2 If pressing the [SETING] key results in a warning message being displayed, retry data entry, or press the [CANCEL] key to clear the warning message. Note that pressing the reset key does not clear the warning message.

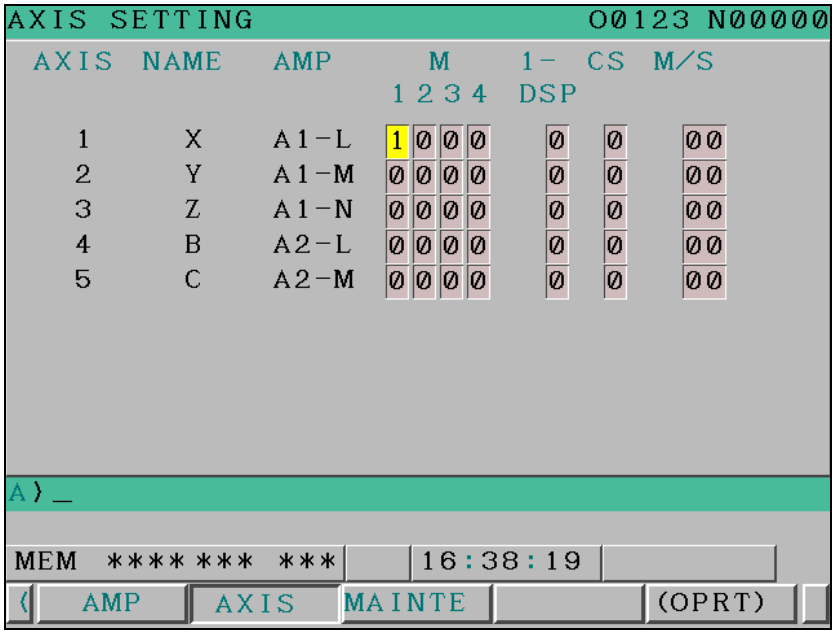
(1) Amplifier setting screen



The amplifier setting screen displays the following items:

- AXIS: Controlled axis number
  For this item, enter a value of between 1 and the maximum number of controlled axes. If a number that falls outside this range is entered, the warning message FORMAT ERROR appears. If the entered controlled axis number is duplicate or 0, the warning message DATA IS OUT OF RANGE appears when the [SETING] soft key is pressed to assert the entered value. In this case, no value can be entered for the parameter.

(2) Axis setting screen



On the axis setting screen, the following items can be specified:

- M1: Connector number for separate detector interface unit 1
  For an axis that uses separate detector interface unit 1, enter a connector number using a number in the range of between 1 and the maximum number of axes for separate detector interface unit 1.
  When separate detector interface unit 1 need not be used, enter 0.
  If a number that falls outside the valid range is entered, the message FORMAT ERROR is displayed.
- M2: Connector number for separate detector interface unit 2
  For an axis that uses separate detector interface unit 2, enter a connector number using a number in the range of between 1 and the maximum number of axes for separate detector interface unit 2.
  When separate detector interface unit 2 need not be used, enter 0.
  If a number that falls outside the valid range is entered, the message FORMAT ERROR is displayed.
- 1–DSP
  Enter 1 for the following axes, each of which exclusively uses a DSP, which is usually shared by two axes. If a number other than 0 or 1 is entered, the message FORMAT ERROR is displayed.
  - High–speed current loop axis
  - High–speed interface axis
- CS: Cs contour controlled axis
  Enter 1 for the Cs contour controlled axis. If a number other than 0 or 1 is entered, the message FORMAT ERROR is displayed.
- TNDM
  Enter odd and even numbers for the master and slave axes for tandem control. These numbers must be consecutive and in the range of between 1 and 4. If a number that falls outside the valid range is entered, the message FORMAT ERROR is displayed.
  This item is not used in Series 0*i* Mate-D.

When the [SETING] soft key is pressed on the axis setting screen after data entry, the message DATA IS OUT OF RANGE is displayed if any of the following conditions is satisfied.

- Both M1 and M2 are nonzero for an axis.
- Any two of the 1–DSP, CS, and TNDM are nonzero for an axis.
- A duplicate value is specified for M1.
- A duplicate value is specified for M2.
- A duplicate value is specified for CS.
- A duplicate value is specified for TNDM.
- An invalid master/slave axis pair is specified for TNDM.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
14476								DFS

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 DFS The FSSB enters:
0: The FS0*i*-D-specific mode.
1: The FS0*i*-C-compatible mode.

	#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 bit 0 (DFS) of parameter No. 14476 is 0:
If the relationship between the axis and the amplifier and the like are defined on the FSSB setting screen, parameters No. 1023, No. 1905, Nos. 1936 and 1937, Nos. 14340 to 14357, and Nos. 14376 to 14391 are automatically set.)
(When bit 0 (DFS) of parameter No. 14476 is 1:
If the relationship between the axis and the amplifier and the like are defined on the FSSB setting screen, parameters No. 1023, No. 1905, Nos. 1910 to 1919, and Nos. 1936 and 1937 are automatically set.)
1: Manual setting 2 mode.
(When bit 0 (DFS) of parameter No. 14476 is 0:
Manually set parameters No.1023, No.1905, Nos.1936 and 1937, Nos.14340 to 14357, and Nos.14376 to 14391.)
(When bit 0 (DFS) of parameter No. 14476 is 1:
Manually set parameters No.1023, No.1905, Nos.1910 to 1919, and Nos. 1936 and 1937.)

#1 ASE When automatic setting mode is selected for FSSB setting (when the FMD parameter (bit 0 of 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.

	#7	#6	#5	#4	#3	#2	#1	#0
1904								DSPx

[Input type] Parameter input
[Data type] Bit axis

#0 DSPx One DSP is used by:
0: Two axes.
1: One axis exclusively.
Since this parameter is set on the FSSB setting screen, do not set it directly. In the manual setting 2 mode, this parameter does not need to be set.

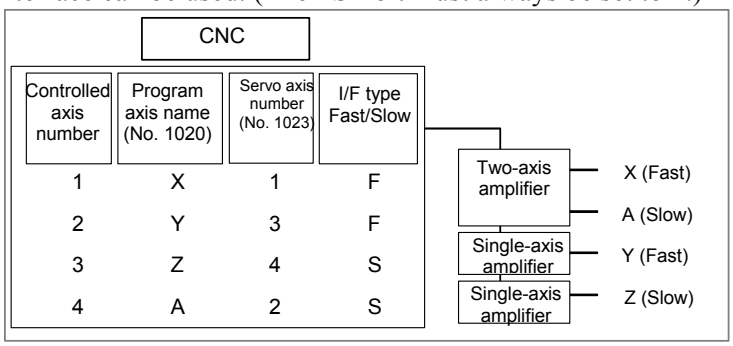
	#7	#6	#5	#4	#3	#2	#1	#0
1905	PM2x	PM1x						FSLx

[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 FSLx The type of interface used between the servo amplifier and servo software is:
0 : Fast type.
1 : Slow type.
The user can choose between two interface types for servo data transfer: fast type or slow type. Set this parameter so that the following conditions are satisfied:

- When a one-axis amplifier is used, either the fast type or slow type interface can be used.
- When a two-axis amplifier is used, the use of the fast type for both axes is not allowed. The slow type can be used for both axes.
- When a three-axis amplifier is used, the requirement for a two-axes amplifier described above applies to the first and second axes, and the requirement for a one-axis amplifier, again described above, applies to the third axis.
- When an odd number is specified for parameter No.1023, the fast type interface must be used. However, the slow type may be used for high-speed current loop axis and high-speed interface axis.
- When an even number is specified for parameter No.1023, only the slow type interface can be used. (The FSL bit must always be set to 1.)



#6 PM1x The first separate detector interface unit is:
0: Not used.
1: Used.

#7 PM2x The second separate detector interface unit is:
0: Not used.
1: Used.

NOTE
When automatic setting mode is selected for FSSB setting (when the parameter FMD (No.1902#0) is set to 0), this parameter is automatically set when input is performed with the FSSB setting screen. When manual setting 2 mode is selected for FSSB setting (when the parameter FMD (No.1902#0) is set to 1), this parameter must be set directly. When a separate detector interface unit is used, a connector number must be set in the corresponding parameter (No.1936 or No.1937).

1910	Address conversion table value for slave 1 (ATR)
1911	Address conversion table value for slave 2 (ATR)
1912	Address conversion table value for slave 3 (ATR)
1913	Address conversion table value for slave 4 (ATR)
1914	Address conversion table value for slave 5 (ATR)
1915	Address conversion table value for slave 6 (ATR)
1916	Address conversion table value for slave 7 (ATR)
1917	Address conversion table value for slave 8 (ATR)
1918	Address conversion table value for slave 9 (ATR)
1919	Address conversion table value for slave 10 (ATR)

NOTE
When these parameters are set, the power must be turned off before operation is continued.

[Input type] Byte
[Valid data range] 0 to 3, 16, 40, 48

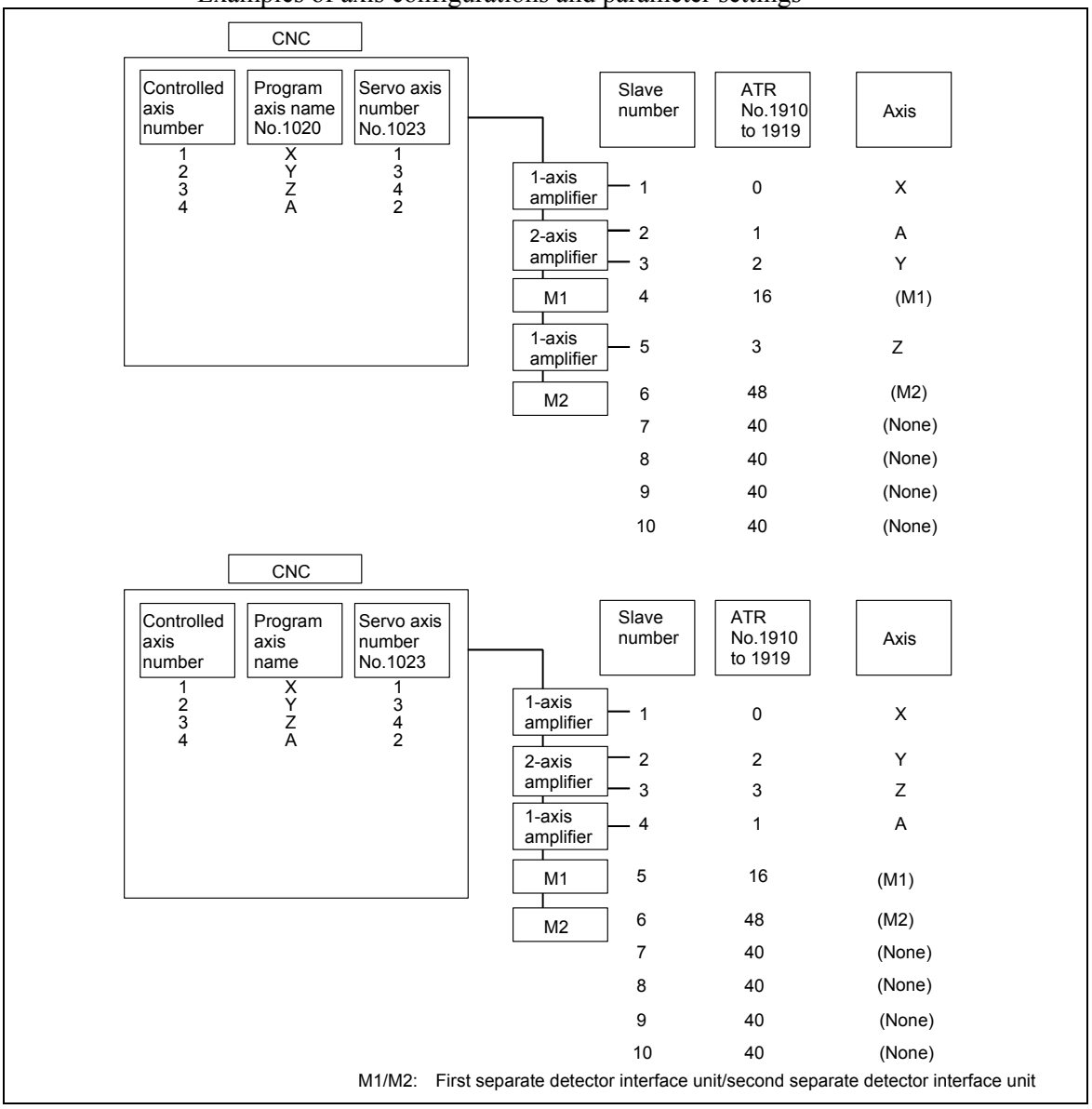
These parameters set address conversion table values for slaves 1 to 10.
A slave is the generic name given to a device such as a servo amplifier or separate detector interface unit, connected to the CNC via an FSSB optical cable. Smaller numbers, starting from 1 are assigned to slaves closer to the CNC; the maximum number that can be assigned is 10. A two-axis amplifier has two slaves, while a three-axis amplifier has three slaves. Set each parameter as described below, depending on whether the slave is an amplifier or separate detector interface unit, or when no slave exists.

- When the slave is an amplifier:
  Set the value obtained by subtracting 1 from the setting of parameter No.1023 for the axis to which the amplifier is assigned.
- When the slave is a separate detector interface unit:
  Set 16 for the first separate detector interface unit (closest to the CNC).
  Set 48 for the second separate detector interface unit (furthest from the CNC).
- When no slave exists
  Set 40.

NOTE
1 When using the simple electric gear box (EGB) function
The EGB axis (axis set with parameter No.7771) does not actually require an amplifier. So, assume that the EGB axis is connected to a dummy amplifier. Accordingly, as the address conversion table value for a nonexistent slave, set the value obtained by subtracting 1 from the setting made for parameter No.1023 for the EGB axis, instead of setting 40.

NOTE
2 When automatic setting mode is selected for FSSB setting (when bit 0 (FMD) of parameter No.1902 is set to 0), parameters No.1910 to No.1919 are automatically set when input is performed with the FSSB setting screen. When manual setting 2 mode is selected for FSSB setting (when bit 0 (FMD) of parameter No.1902 is set to 1), parameter No.1910 to No.1919 must be directly set.

- Examples of axis configurations and parameter settings



1936	Connector number of the first separate detector interface unit

1937	Connector number of the second 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 7

This parameter sets the connector number corresponding to the connector connected when using the separator detector interface unit set by bits 6 and 7 of parameter No. 1905 minus 1. That is, set 0 to 7 for connector numbers 1 to 8, respectively. Set 0 for the axis for which the separator detector interface unit is not used.
Use successive numbers for one separator detector interface unit. Do not omit a intermediate number.

Example)

Controlled axis	Connector number for the first separate detector interface unit	Connector number for the second separate detector interface unit	No.1936	No.1937	PM2x, PM1x (No.1905#7, #6)
X	1	Not used	0	0	0, 1
Y	Not used	2	0	1	1, 0
Z	Not used	1	0	0	1, 0
A	Not used	Not used	0	0	0, 0

NOTE
When automatic setting mode is selected for FSSB setting (when the parameter FMD (No.1902#0) is set to 0), these parameters are automatically set when input is performed with the FSSB setting screen. When manual setting 2 mode is selected for FSSB setting (when the parameter FMD (No.1902#0) is set to 1), these parameters must be set directly.

Reference item

Manual name	Item name
CONNECTION MANUAL (HARDWARE) (B-64303EN)	Connection to the servo amplifiers

1.4.4 Temporary Absolute Coordinate Setting

Overview

In the full closed system with an inner absolute position pulse coder (serial pulse coder) and an incremental scale, the position is set by using absolute position data from the inner absolute position pulse coder at the power on sequence. After that, the position is controlled with incremental data from the incremental scale. The position just after power on sequence is rough, and the manual reference position return is required to get the accurate position.
With this function, the position at the power on is rough, but the following functions are available before the reference position return.

- Stroke limit check
- Position switch

Please be careful this function does not make an incremental scale work as an absolute position detector.

NOTE
Temporary absolute coordinate setting function is optional function.

In using the absolute position detector by inductosyn of the axis, this function is not available.

Also in using the linear-scale with distance-coded reference marks(serial), these function are not available.

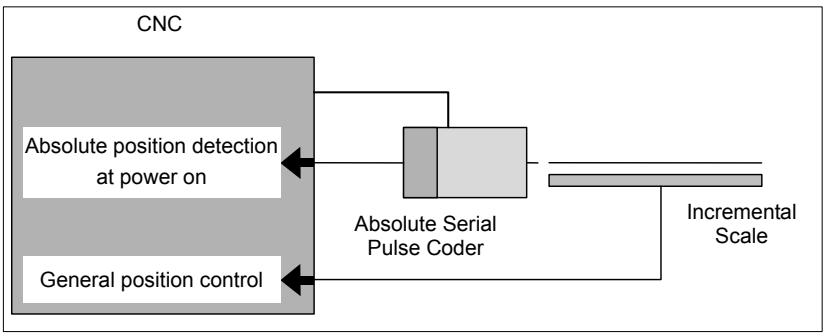


Fig. 1.4.4 (a) The system with the Temporary Absolute Coordinate Setting

In using the temporary absolute coordinate, the coordinate is same as the reference position return has not been performed.
The reference position return is required to get the accurate position after the power on sequence.

The reference position establishment signal <Fn120.0 to Fn120.4> are not set to "1" until the manual reference point return is performed.
The diagnosis No.304 (reference counter value) are also not displayed until the manual reference point return is performed.

Parameters

	#7	#6	#5	#4	#3	#2	#1	#0
1815			APCx	APZx			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 pulse coder is not used.
1: A separate pulse coder is used.

NOTE
In case of using this function, this parameter is set to "1".

#4 APZx Machine position and position on absolute position detector when the absolute position detector is used
0 : Not corresponding
1 : Corresponding

NOTE
1 If the following parameters is modified, the parameter APZ (No.1815#4) will be changed to "0".
No.1815#1, No.1820, No.1821, No.1874, No.1875, No.2022, No.2084, and No.2085
2 The parameter APZ (No.1815#4) is kept value "1" even if the grid shift value is modified.

#5 APCx Position detector is
0 : other than an absolute position detector.
1 : an absolute position detector (absolute pulse coder).

NOTE
In case of using this function, this parameter is set to "1".

1874	Numerator of the conversion coefficient for inner pulse coder
1875	Denominator of the conversion coefficient for inner pulse coder

NOTE
When these parameters are set, the power must be turned off before operation is continued..

[Input type] Parameter input
[Data type] word axis
[Unit of data]
[Valid data range] 1 to 32767
Set a conversion coefficient for the Temporary Absolute Coordinate Setting for each axis. The value set is determined as follows :

$$\frac{\text{No.1874}}{\text{No.1875}} = \frac{\text{Number of position feedback pulses per motor revolution}}{\text{1,000,000}}$$

	#7	#6	#5	#4	#3	#2	#1	#0
2011	XIAx							

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#7 XIAx The Temporary Absolute Coordinate Setting function is
0 : invalid.
1 : valid.

NOTE
1 In case of using the Temporary Absolute Coordinate Setting, parameters No.1815#1(OPTx), No.1815#5(APCx), No.1874 and No.1875 must be set.
2 For the setting of this parameter to take effect, the power must be turned off and back on again.

1.5 SETTINGS RELATED WITH COORDINATE SYSTEMS

1.5.1 Machine Coordinate System

Overview

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. A machine coordinate system, once set, remains unchanged until the power is turned off.

Format

M

(G90)G53 IP_;
IP_: Absolute dimension word

T

G53 IP_;
IP_: Absolute dimension word

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 rapid traverse block overlap function between a machine coordinate system selection command (G53) block and the positioning (rapid traverse) command (G00) block, allowing the tool to execute the next rapid traverse command (G00) without slowing down or stopping at the end of the machine coordinate system selection command (G53). This ensures high-speed positioning even when the machine coordinate system selection command (G53) is used.
Specifying P1 in the G53 block enables the high-speed G53 function.

Format

M

(G90)G53 IP_P1;
IP_; Absolute dimension word
P1; The high-speed G53 function is enabled.

T

G53 IP _P1 ;
IP_; Absolute dimension word
P1; The high-speed G53 function is enabled.

Limitation

- Cancel of the compensation function

When the G53 command is specified, cancel the compensation functions such as the cutter compensation, tool length offset, tool-nose radius compensation, and tool offset.

- G53 specification immediately after power-on

Since the machine coordinate system must be set before the G53 command is specified, at least one manual reference position return or automatic reference position return by the G28 command must be performed after the power is turned on. This is not necessary when an absolute-position detector is attached.

- Blocks for which the high-speed G53 function takes effect

The high-speed G53 function takes effect for the following combinations of commands:

- G53→G00
- G53→G53

The high-speed G53 function does not take effect for the following combination of commands:

- G00→G53

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.

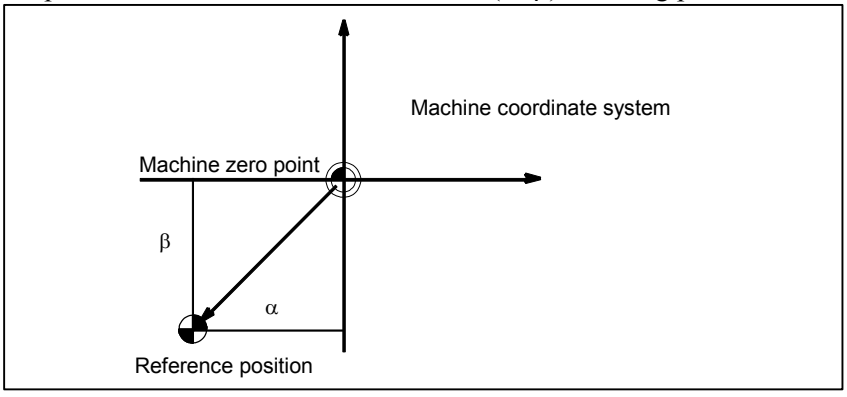
T

Commands G50/G51 (except for system A), G50.1/G51.1, and G68.1/G69.1 cannot be specified in the same block where the G53 command is specified.

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.



Parameter

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.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Machine coordinate system

1.5.2 Workpiece Coordinate System/Addition of Workpiece Coordinate System Pair

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).

1.5.2.1 Workpiece coordinate system

NOTE
To use the workpiece coordinate system, set 0 in bit 0 (NWZ) of parameter No. 8136.

- 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 (M series :G92, T series :G50 (G92 when the G code series is B or C)
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.
Note that this is invalid when the workpiece coordinate system is provided (when 0 is set in bit 0 (NWZ) of parameter No. 8136). In that case, the workpiece coordinate system is always established based on the workpiece zero point offset value (parameter Nos. 1220 to 1226), regardless of the setting of ZPR, when manual reference position return is performed.

(3) Method using a workpiece coordinate system selection G code
Six workpiece coordinate systems can be set beforehand using the MDI panel. Program commands G54 to G59 can be used to select the work axis to be used.

When using an absolute command, establish the workpiece coordinate system in any of the above ways.

⚠ CAUTION
The established workpiece coordinate system depends on diameter programming or radius programming.

- Selecting a workpiece coordinate system

The user can choose from set workpiece coordinate systems as described below.

(1) Once a workpiece coordinate system is set using a workpiece coordinate system setting G code [M series :G92, T series :G50 (G92 when the G code series is B or C)] or automatic setting, all subsequent absolute commands use positions in this workpiece coordinate system.

(2) Choosing from six workpiece coordinate systems set using the MDI panel
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 as default.

When bit 2 (G92) of parameter No. 1202 is set to 1, executing the G code command for coordinate system setting (G92 <M series>, G50 <Tseries> (G92 with G code system B/C for T series)) results in the issue of alarm (PS0010). This is designed to prevent the user from confusing coordinate systems.

> **⚠ CAUTION**
> The set workpiece zero point offset value depends on diameter programming or radius programming.

- Changing workpiece coordinate system

The six workpiece coordinate systems specified with G54 to G59 can be changed by changing an external workpiece zero point offset value or workpiece zero point offset value.
Three methods are available to change an external workpiece zero point offset value or workpiece zero point offset value.

(1) Inputting from the MDI panel

(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 zero point offset value can be changed by input signal to CNC. Refer to machine tool builder's manual for details.

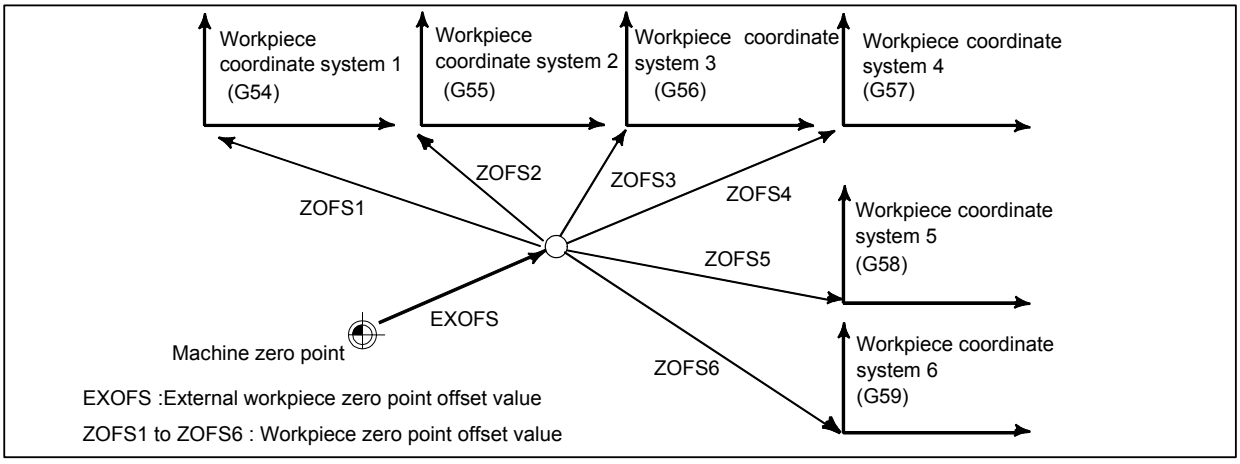


Fig. 1.5.2 (a) Changing an external workpiece zero point offset value or workpiece zero point offset value

Format

- Changing by inputting programmable data

G10 L2 Pp IP_;

p=0 : External workpiece zero point offset value

p=1 to 6: Workpiece zero point offset value correspond to workpiece coordinate system 1 to 6

IP_: For an absolute command, workpiece zero point offset for each axis.
For an incremental command, value to be added to the set workpiece zero point offset for each axis (the result of addition becomes the new workpiece zero point 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 zero point 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 zero point offset values. This means that all the workpiece coordinate systems are shifted by the same amount.

⚠ CAUTION
When a coordinate system is set with G code command for workpiece coordinate system setting after an external workpiece zero point offset value is set, the coordinate system is not affected by the external workpiece zero point 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.

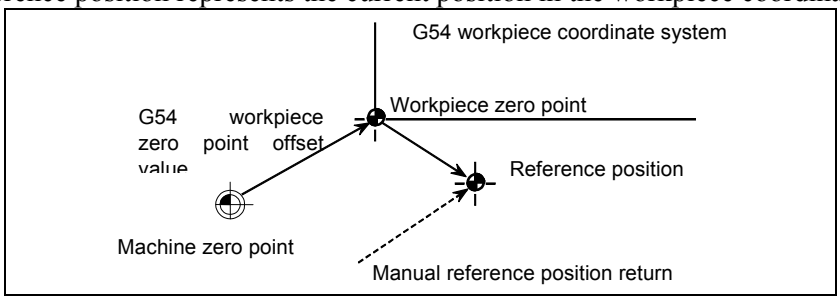
T

If IP is an incremental command value, the work 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)

1.5.2.2 Workpiece coordinate system preset

NOTE
To use the workpiece coordinate system preset function, set 0 in bit 1 (NWC) of parameter No. 8136.

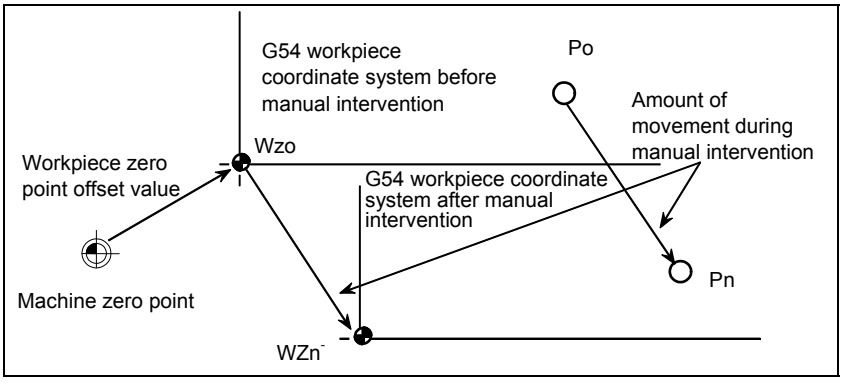
Reset state, a workpiece coordinate system is shifted by the workpiece zero point offset value from the machine coordinate system zero point. Suppose that the manual reference position return operation is performed when a workpiece coordinate system is selected with G54. In this case, a workpiece coordinate system is automatically set which has its zero point displaced from the machine zero point by the G54 workpiece zero point offset value; the distance from the zero point 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 zero point displaced from the machine zero point by the G54 workpiece zero point 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 zero point 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 interrupt
(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.



A workpiece coordinate system shifted through the above operation can be preset to a workpiece coordinate system offset by the workpiece zero point offset value from the machine zero point, as when the manual reference position return operation is performed, by using a G code command [M series :G92.1, T series :G50.3 (G92.1 when the G code series is B or C)] or the MDI panel.
Whether to preset relative coordinates (RELATIVE) as well as absolute coordinates can be chosen using bit 3 (PPD) of parameter No. 3104.

When a workpiece coordinate system (G54 to G59) is not provided (1 is set in bit 0 (NWZ) of parameter No. 8136), the workpiece coordinate system is preset to the coordinate system whose reference point is the zero point.

Limitation

M

- Cutter compensation, tool length compensation, tool offset

When using the workpiece coordinate system preset function, cancel compensation modes: cutter compensation, tool length compensation, and tool offset. If the function is executed without cancelling these modes, compensation vectors are cancelled.

T

- Tool-nose radius compensation, tool offset

When using the workpiece coordinate system preset function, cancel compensation modes: tool-nose radius compensation and tool offset. If the function is executed without cancelling these modes, compensation vectors are cancelled.

- Program restart

The workpiece coordinate system preset function is not executed during program restart.

- Prohibited modes

Do not execute the workpiece coordinate system preset function in the scaling (M series), coordinate system rotation (M series), or programmable mirror image (M series) mode.

1.5.2.3 Adding workpiece coordinate systems (G54.1 or G54) (M series)

Besides the six workpiece coordinate systems (standard workpiece coordinate systems) selectable with G54 to G59, 48 additional workpiece coordinate systems (additional workpiece coordinate systems) can be used.

NOTE
To use additional workpiece coordinate systems, set 1 in bit 2 (NWN) of parameter No. 8136.

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).
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

As with the standard workpiece coordinate systems, the following operations can be performed for a workpiece zero point offset in an additional workpiece coordinate system:

(1) The workpiece zero point offset value setting screen can be used to display and set a workpiece zero point offset value.
(2) The G10 function enables a workpiece zero point offset value to be set by programming.

(3) A custom macro allows a workpiece zero point offset value to be handled as a system variable.
(4) Workpiece zero point offset data can be entered or output as external data.
(5) The PMC window function enables workpiece zero point offset data to be read as program command modal data.

- Setting the workpiece zero point offset value in the additional coordinate systems (G10)

When a workpiece zero point 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;

1.5.2.4 Automatic coordinate system setting

When ZPR (bit 0 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:
Note that this cannot be used when the workpiece coordinate system is provided (when 0 is set in bit 0 (NWZ) of parameter No. 8136). In that case, the workpiece coordinate system is always established based on the workpiece zero point offset value (parameter Nos. 1220 to 1226), regardless of the setting of ZPR, when manual reference position return is performed.

M

G92 Xα Yβ Zγ ;

T

G50 Xα Zγ ;

When the setting of a workpiece coordinate system shift amount is other than 0, a workpiece coordinate system shifted by the amount is set.

1.5.2.5 Workpiece coordinate system shift (T series)

T

Explanation

When the coordinate system actually set by the G50 command or the automatic system setting deviates from the programmed work system, the set coordinate system can be shifted. Set the desired shift amount in the work coordinate system shift memory.

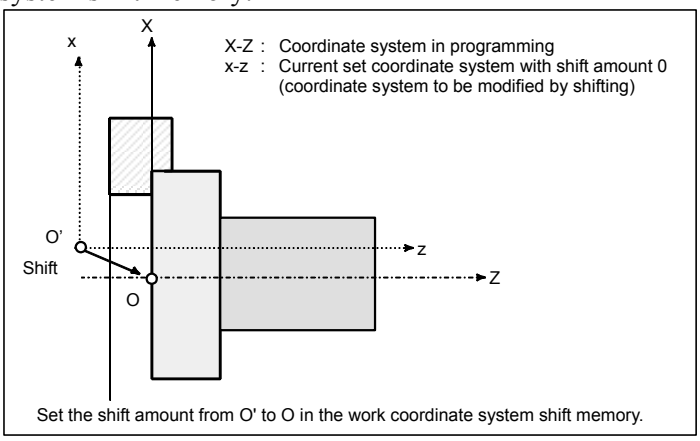


Fig. 1.5.2 (b) Workpiece coordinate system shift

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.

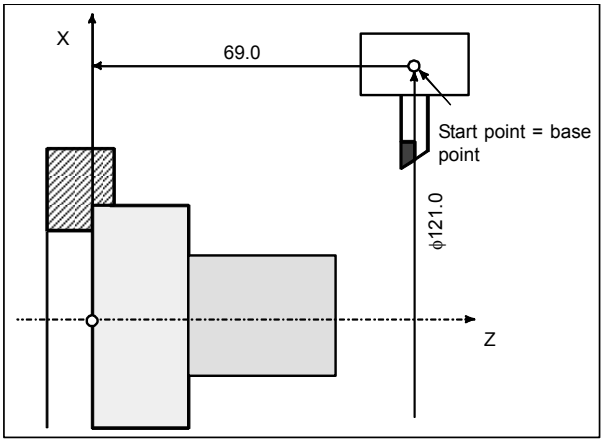
- Diameter and radius values

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 zero point, the actual position is at X = ϕ121.0 and Z = 69.0 from the zero point. Set a shift amount as shown below:

X=1.0, Z=-1.0



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1201	WZR	NWS						ZPR
	WZR							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 No. 1220 to No. 1226), irrespective of this parameter setting.

#6 NWS The workpiece coordinate system shift amount setting screen is:
0: Displayed
1: Not displayed

NOTE
When the workpiece coordinate shift amount setting screen is not displayed, a workpiece coordinate system shift amount modification using G10P0 cannot be made.

#7 WZR If the CNC is reset by the reset key on the MDI panel, external reset signal, reset and rewind signal, or emergency stop signal when bit 6 (CLR) of parameter No. 3402 is set to 0, the G code of group number 14 (workpiece coordinate system) is:
0: Placed in the reset state
1: Not placed in the reset state

NOTE
When bit 6 (CLR) of parameter No. 3402 is set to 1, whether to place the G code in the reset state depends on bit 6 (C14) of parameter No. 3407.

	#7	#6	#5	#4	#3	#2	#1	#0
1202						G92	EWS	EWD
						G92		EWD

[Input type] Parameter input
[Data type] Bit path

#0 EWD The shift direction of the workpiece coordinate system is:
0: The direction specified by the external workpiece zero point offset value
1: In the opposite direction to that specified by the external workpiece zero point offset value

#1 EWS The external workpiece zero point offset is made:
0: Valid
1: Invalid

NOTE
When the external workpiece zero point offset is made invalid, the following operation results:
1 As the external workpiece zero point offset on the workpiece zero point offset setting screen, a workpiece coordinate system shift amount is displayed.
2 Data keyed through the MDI panel for the workpiece coordinate system shift amount and external workpiece zero point offset is loaded into the memory for the workpiece coordinate system shift amount.
3 A write to or read from the workpiece coordinate system shift amount and external workpiece zero point offset with a macro variable is performed using the respective memory.
4 A write to or read from the workpiece coordinate system shift amount and external workpiece zero point offset with the window function is performed using the respective memory.

#2 G92 When the workpiece coordinate system is used (when bit 0 (NWZ) of parameter No. 8136 is 0), if the G code (M series: G92, T series: G50) for coordinate system setting is specified:
0: G command is executed and no alarm is issued.
1: G command is not executed and an alarm (PS0010) is issued.

1220	External workpiece zero point offset value in each axis

[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 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 is one of the parameters that give the position of the zero point of workpiece coordinate system (G54 to G59). It gives an offset of the workpiece zero point common to all workpiece coordinate systems. In general, the offset varies depending on the workpiece coordinate systems. The value can be set from the PMC using the external data input function.

1221	Workpiece zero point offset value in workpiece coordinate system 1 (G54)
1222	Workpiece zero point offset value in workpiece coordinate system 2(G55)
1223	Workpiece zero point offset value in workpiece coordinate system 3(G56)
1224	Workpiece zero point offset value in workpiece coordinate system 4 (G57)
1225	Workpiece zero point offset value in workpiece coordinate system 5 (G58)
1226	Workpiece zero point offset value in workpiece coordinate system 6 (G59)

[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 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)
The workpiece zero point offset values in workpiece coordinate systems 1 to 6 (G54 to G59) are set.

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 setting a coordinate system automatically.

	#7	#6	#5	#4	#3	#2	#1	#0
3104					PPD			

[Input type] Parameter input
[Data type] Bit path

#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 lath system)
(4) T code command for the T series

	#7	#6	#5	#4	#3	#2	#1	#0
3290				IWZ	WZO			

[Input type] Parameter input
[Data type] Bit path

#3 WZO Setting a workpiece zero point offset value and workpiece shift value (T series) by MDI key input is:
0: Not disabled
1: Disabled

#4 IWZ Setting a workpiece zero point offset value or workpiece shift value (T series) by MDI key input in the automatic operation activation or halt state is:
0: Not disabled
1: Disabled

1.5.2.6 Each axis workpiece coordinate system preset signals

Overview

The each axis workpiece coordinate system preset signals are fuctions for presetting a workpiece coordinate system shifted due to manual intervention, a machine lock, etc. to a workpiece coordinate system offset from the pre-shift machine zero point by a workpiece origin offset value, using an input signal.

NOTE
To use the each axis workpiece coordinate system preset signals, set 1 in bit 6 (WPS) of parameter No. 3006.

Signal

Each axis workpiece coordinate system preset signals WPRST1 to WPRST5 <Gn358.0 to Gn358.4>

[Classification] Input signal
[Function] These signals are used to preset a workpiece coordinate system shifted due to manual intervention, a machine lock, etc.
[Operation] By changing the signal for the axis for which to perform a workpiece coordinate system preset from "0" to "1", the workpiece coordinate system preset is performed. This cancels the shift amount for the workpiece coordinate system that is due to any of the items below, so that the workpiece coordinate system can be preset to a workpiece coordinate system preset from the pre-shift machine zero point by the workpiece origin offset value.

(a) Manual intervention performed when the manual absolute signal is off

(b) Move command executed in the machine lock state
(c) Movement by handle interrupt
(d) Operation using the mirror image function
(e) Setting of a local coordinate system with G52 or shift of a workpiece coordinate system with G92/G50 (for the T series)

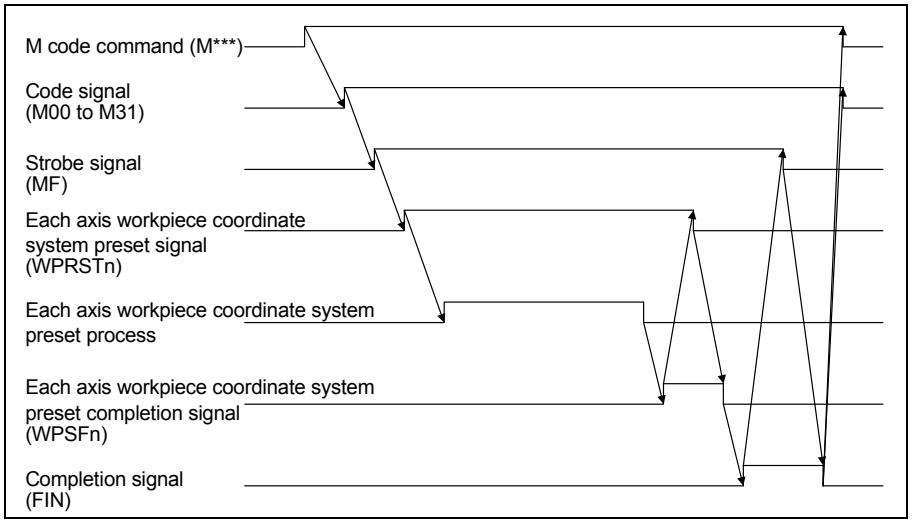
NOTE
1 If performing workpiece coordinate system preset with an each axis workpiece coordinate system preset signal during automatic operation, specify it with the M code set in parameters Nos. 11275 and 11276 or perform it during a single block stop.
2 If preset is to be performed with an each axis workpiece coordinate system preset signal during automatic operation, the state must be that in which all axes on the path including the axis on which to perform each axis workpiece coordinate system preset are stopped and no commands are in execution. If not all axes are stopped or if a command is in execution, alarm PS1820 is issued.
3 When an M code for performing each axis workpiece coordinate system preset is issued during automatic operation, if the corresponding each axis workpiece coordinate system preset signal is not set to ”1”, alarm PS1820 is issued. It is possible to suppress the alarm by setting bit 0 (WPA) of parameter No. 11277 to 1. In this case, axis workpiece coordinate system preset is not performed.
4 During a reset (RST<Fn001.1> = “1”), preset with an each axis workpiece coordinate system preset signal is disabled. Preset is performed at the point the rest is canceled.
5 During an auxiliary function lock, this function is disabled.

Each axis workpiece coordinate system preset completion signals WPSF1 to WPSF5 <Fn358.0 to Fn358.4>

[Classification] Output signal
[Function] These signals notify the PCM of the each axis workpiece coordinate system preset status.
[Output cond.] These signals become “1” in the following case:
- When workpiece coordinate system preset is completed with the corresponding each axis workpiece coordinate system preset signals.

They become “0” in the following cases:
- When the corresponding each axis workpiece coordinate system preset signals change from “1” to “0”.
- When a reset is performed.

- Timing chart



Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn358				WPRST5	WPRST4	WPRST3	WPRST2	WPRST1
Fn358				WPSF5	WPSF4	WPSF3	WPSF2	WPSF1

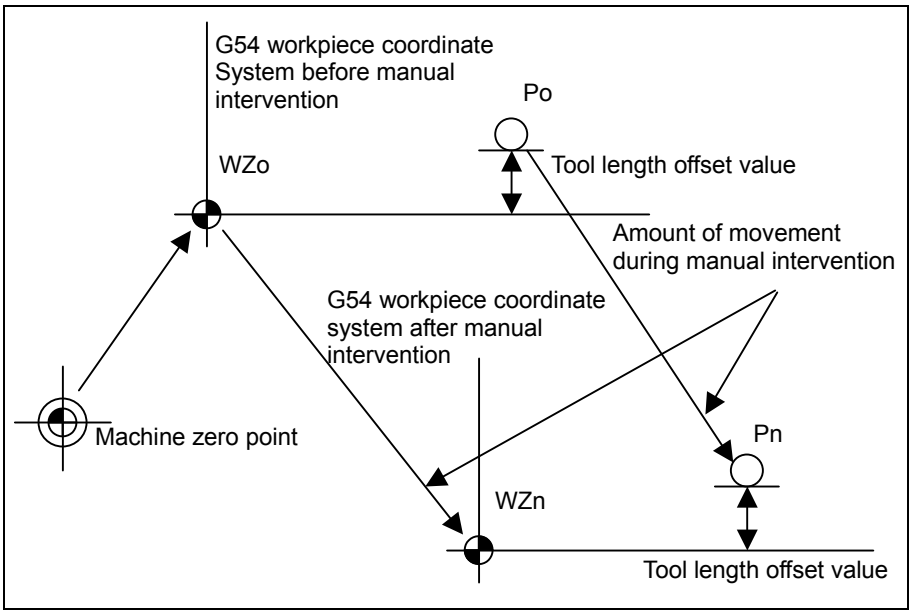
Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1205								
	WTC							

[Input type] Parameter input
[Data type] Bit path

#7 WTC When workpiece coordinate system preset is done, actual tool length offset is:
0: Not considered.
1: Considered.

When this paramter is set 1, it is possible to preset the workpiece coordinate system by G-code, MDI operation or the workpiece coordinate system preset signal without cancelling the tool length compensation modes.
The compensation vector is kept as the below figure when the workpiece coordinate system preset is done to the coordinate shifted by amount of movement during manual intervention.



	#7	#6	#5	#4	#3	#2	#1	#0
3006		WPS						

[Input type] Parameter input
[Data type] Bit

#6 WPS Each axis workpiece coordinate system preset signal:
0: Disabled.
1: Enabled.
When this parameter is set to 1, a workpiece coordinate system is preset after the end of the high speed program check mode.

11275	The top number of M code used to turn on each axis workpiece coordinate system preset signal

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 99999999
Specify the top number of M code for turning "1" each axis workpiece coordinate system preset signal (Gn358.*) during automatic operation.
When the specified M codes are within the range specified with this parameter and parameter No.11276, each axis workpiece coordinate system preset signal is checked and preset workpiece coordinate system for axis that the signal is turned "1".
The specified M codes prevent buffering.

NOTE
When each axis workpiece coordinate system preset signals are turned "1" more than two signals by an M code, please turn "1" the signals of all axis at the same timing. If the timing is different, only the axis of the first signal turned "1" is preset.
If you want to turn "1" the signals at the different timing, please specify M code separately.

11276	The number of M code used to turn on each axis workpiece coordinate system preset signal

[Input type] Parameter input

[Data type] Word path
[Valid data range] 1 to 999

Specify the number of M code for turning "1" each axis workpiece coordinate system preset signal (Gn358.*) during automatic operation.
For example, when parameter No.11275 = 100 and parameter No.11276 = 10 are set, From M100 to M109 are used for turning "1" each axis workpiece coordinate system preset signal.
When 0 is set, the number of M code is assumed to be 1.

NOTE
Set only M code that is not used for another function. (M00 to 05,30,98,99, M code used to call the subprogram, etc.)

	#7	#6	#5	#4	#3	#2	#1	#0
11277								WPA

[Input type] Parameter input
[Data type] Bit path

#0 WPA When an M code for turning on the workpiece coordinate system preset signal for an axis is specified, but the signal is not turned on, or an auxiliary function lock is provided:
0: An alarm (PS1820) is issued.
1: An alarm is not issued.

Alarm and message

Number	Message	Description
PS1820	ILLEGAL DI SIGNAL STATE	1. An each axis workpiece coordinate system preset signal was turned on 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 on. 3. The auxiliary function lock is enabled.

Notes

NOTE
The limitations are the same as those for the workpiece coordinate system preset using a program command (G92.1 or G50.3 (for G code system A (T series) or using MDI operation. Thus, before performing preset with this function, cancel each compensation mode (cutter compensation, tool offset, and tool length compensation). Otherwise, the compensation vector will be canceled. If this occurs, speciy each compensation mode again. For details, refer to the Operator's Manual (Common to Lathe System/Machining Center System) (B-64304EN).

1.5.3 Local Coordinate System

Overview

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 zero point of a new local coordinate system in the workpiece coordinate system.

To cancel the local coordinate system and specify the coordinate value in the workpiece coordinate system, match the zero point of the local coordinate system with that of the workpiece coordinate system.

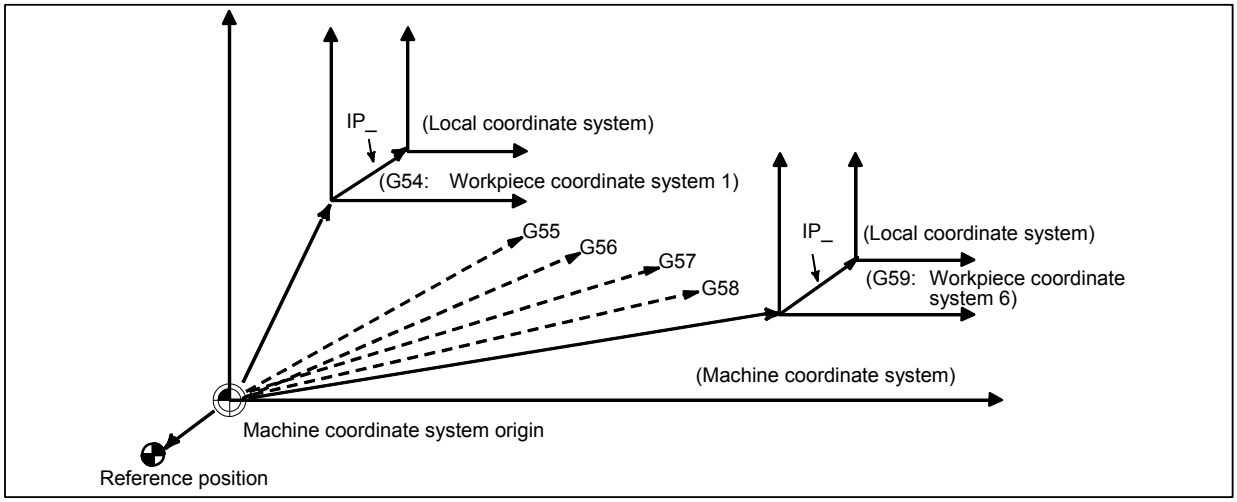


Fig. 1.5.3 (a) Setting the local coordinate system

⚠ CAUTION

1 When ZCL (bit 2 of parameter No.1201) is set to 1 and an axis returns to the reference point by the manual reference point return function, the zero point of the local coordinate system of the axis matches that of the work coordinate system.
The same is true when the following command is issued:
G52α0;
α: Axis which returns to the reference point
2 The local coordinate system setting does not change the workpiece and machine coordinate systems.
3 Whether the local coordinate system is canceled upon reset depends on the specified parameters. The local coordinate system is canceled upon reset when CLR (bit 6 of parameter No. 3402) or RLC (bit 3 of parameter No. 1202) is set to 1.

⚠ CAUTION

4 When a workpiece coordinate system is set with the G code command for coordinate system setting (G92 <M series> , G50 <T series> (G92 with G code system B/C for T series)), the local coordinate system is canceled. However, the local coordinate system of an axis for which no coordinate system is specified in a G code block for coordinate system setting remains unchanged.
5 G52 cancels the offset temporarily in cutter or tool-nose radius compensation.
6 Command a move command immediately after the G52 block in the absolute mode.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1201						ZCL		

[Input type] Parameter input
[Data type] Bit path

#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 is provided (when 0 is set in bit 0 (NWZ) of parameter No. 8136). In order to use the local coordinate system (G52), the workpiece coordinate system is required.

	#7	#6	#5	#4	#3	#2	#1	#0
1202					RLC			

#3 RLC Local coordinate system is
0: Not cancelled by reset
1: Cancelled by reset

NOTE
1 When bit 6 (CLR) of parameter No. 3402 is set to 0, and bit 7 (WZR) of parameter No. 1201 is set to 1, the local coordinate system is cancelled, regardless of the setting of this parameter.
2 When bit 6 (CLR) of parameter No. 3402 is set to 1, and bit 6 (C14) of parameter No. 3407 is set to 0, the local coordinate system is cancelled, regardless of the setting of this parameter.

1.5.4 Rotary Axis Roll Over

Overview

The roll-over function prevents coordinates for the rotation axis from overflowing. The roll-over function is enabled by setting parameter ROAx (No. 1008#0) to 1.

For an incremental command, the tool moves the angle specified in the command. For an absolute command, the coordinates after the tool has moved are values rounded by the angle corresponding to one rotation set in parameter No. 1260. The tool moves in the direction in which the final coordinates are closest when parameter RABx (No. 1008#1) is set to 0. Displayed values for relative coordinates are also rounded by the angle corresponding to one rotation when parameter RRLx (No. 1008#2) is set to 1.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1006							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 rotation axis.
#1 ROSx Setting linear or rotation 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	Rotation axis (A type) (1) Inch/metric conversion is not done. Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by parameter No.1008#0(ROAx) and #2(RRLx). (2) Stored pitch error compensation is the rotation type. (Refer to parameter No.3624) (3) 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	Rotation axis (B type) (1) Inch/metric conversion, absolute coordinate values and relative coordinate values are 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 rotation axis roll over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

	#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 roll-over function of a rotation axis is
0: Invalid
1: Valid

NOTE
ROAx specifies the function only for a rotation axis (for which ROTx, #0 of parameter No.1006, is set to 1)

#1 RABx In the absolute commands, 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.

1260	Amount of a shift per one rotation of a rotation 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 amount of a shift per one rotation of a rotation axis.
For the rotation axis used for cylindrical interpolation, set the standard value.

Note

NOTE
This function cannot be used together with the indexing function of the index table (M series).

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Rotary axis roll-over function

1.6 AXIS SYNCHRONOUS CONTROL

Overview

When a movement is made along one axis by using two servo motors as in the case of a large gantry machine, a command for one axis can drive the two motors by synchronizing one motor with the other.
An axis used as the reference for axis synchronous control is referred to as a master axis (M-axis), and an axis along which a movement is made in synchronism with the master axis is referred to as a slave axis (S-axis).

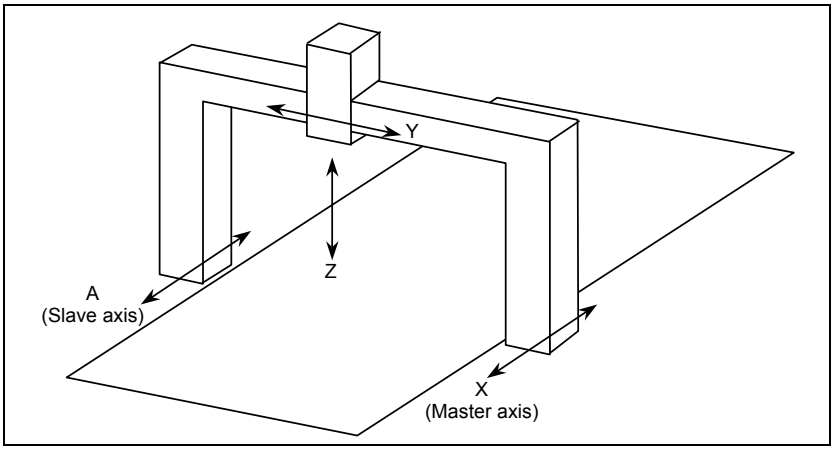


Fig. 1.6 (a) Example of machine with X and A being synchronous axes

The synchronization establishment function automatically performs compensation so as to eliminate the deviation of machine coordinates upon emergency stop cancellation or some other occasion.
An external signal can be used to turn synchronization on and off. When synchronization is turned on and off using an external signal.

1.6.1 Axis Configuration for Axis Synchronous Control

Explanation

- Master axis and slave axis for axis synchronous control

An axis used as the reference for axis synchronous control is referred to as a master axis (M-axis), and an axis along which a movement is made in synchronism with the master axis is referred to as a slave axis (S-axis).
By setting the axis number of a master axis in the parameter (No. 8311) of the slave axis, the axis configuration for axis synchronous control is determined.

- Synchronous operation and normal operation

Operation where axis synchronous control is turned on (enabled) to make a movement along the slave axis in synchronism with the master axis is referred to as synchronous operation. Operation where axis synchronous control is turned off (disabled) to make movements along the master axis and slave axis independently of each other is referred to as normal operation.

(Example)

Automatic operation when the master axis is the X-axis and the slave axis is the A-axis
In synchronous operation, movements are made along the X-axis and A-axis according to the programmed command Xxxxx for the master axis.
In normal operation, movements are made along the master axis and slave axis independently of each other 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.

The mode of operation can be switched between synchronous operation and normal operation by an input signal, or synchronous operation can be performed at all times. Which mode to use can be set using bit 5 (SCA) of parameter No. 8304.

Switching between synchronous operation and normal operation by using an input signal

When bit 5 (SCA) of parameter No. 8304 is set to 0 for the slave axis, the signal SYNCx/SYNCJx (with x representing a slave axis number) is used to switch between synchronous operation and normal operation. When SYNCx/SYNCJx = 1, synchronous operation is selected. When SYNCx/SYNCJx = 0, normal operation is selected.
During axis synchronous control, the output signal SYNOx becomes 1.

Setting for using synchronous operation at all times

When bit 5 (SCA) of parameter No. 8304 for the slave axis is set to 1, synchronous operation is performed at all times, regardless of the setting of the signal SYNCx/SYNCJx.

- Synchronous control axis name

The name of a master axis and the name of a slave axis may be the same or may be different from each other.

Restrictions on using the same name for the master axis and slave axis

If the same axis name is assigned to the master axis and slave axis, manual operation only is allowed in normal operation. Automatic operation and manual numeric command cannot be performed.

Setting of an axis name subscript

A subscript can be attached to an axis name like X1, X2, XM, and XS. If the same axis name is used for multiple axes, and a unique subscript is assigned to each of those axes, the axes can be distinguished from each other on the screen display, or which of those axes issued an alarm can be identified.
Set a subscript in parameter No. 3131.

- Setting of multiple slave axes

One master axis can have multiple slave axes.

(Example)

In the example below, movements along the X1-axis and X2-axis are made in synchronism with the XM-axis.

Axis name indication	Controlled axis number	Axis name No. 1020	Subscript No.3131	Master axis number No.8311	Operation
XM	1	88	77	0	
Y	2	89	0	0	
X1	3	88	49	1	A movement is made in synchronism with the XM-axis.
X2	4	88	50	1	A movement is made in synchronism with the XM-axis.

When one master axis has multiple slave axes, synchronization establishment, and synchronization error check are performed for each slave axis independently.

- Combination with tandem control

Tandem control can be used with each of the master and slave axes. The same restriction on axis arrangement as imposed in the case of normal tandem control is imposed. No particular restriction is imposed on axis synchronous control.

- Axis selection on the screen display

On a screen such as the current position display screen, a slave axis is also displayed. The display of a slave axis can be disabled by setting bit 0 (NDP) of parameter No. 3115 to 1 and setting bit 1 (NDA) of parameter No. 3115 to 1.

- Axis selection in actual cutting feedrate display

By setting bit 2 (SAF) of parameter No. 8303 to 1 for a slave axis, the slave axis can be included in an actual cutting feedrate display calculation during synchronous operation.

- Axis synchronous control with an absolute-position detector

When bit 7 (SMA) of parameter No. 8302 is set to 1 to attach an absolute-position detector, and bit 4 (APZ) of parameter No. 1815 for an axis placed in synchronous operation is turned off, APZ for the axis (axes) placed together in synchronous operation is also turned off.

- Slave axis mirror image

By setting parameter No. 8312, a mirror image can be applied to a slave axis placed in synchronous operation. When the mirror image function is enabled, the direction in which the absolute and relative coordinates change is the same as for the machine coordinates.
At this time, synchronization establishment, synchronization error check, and correction mode cannot be used.
The mirror image set by bit 0 (MIR) of parameter No. 0012 cannot be applied to the slave axis. Because this mirror image differs from the mirror image set by parameter MIR, it does not affect input signal MIx <Gn106.0 to Gn106.4> or output signal MMIx <Fn108.0 to Fn108.4>.

- External machine coordinate system shift

Bit 7 (SYE) of parameter No. 8304 can be set to 1 for the slave axis to shift the slave axis by the same amount as specified for the master axis when external machine coordinate system shift is specified by external data input/output for the master axis in synchronous control.

- How to set parameter No. 7310 when using feed axis synchronization

If not switching between synchronous operation and normal operation with signals (SYNC1 to SYNC5 <Gn138.0 to Gn138.4> and SYNCJ1 to SYNCJ5 <Gn140.0 to Gn140.4>) (to be in the synchronous state at all times), set the same data for the master axis and slave axis.
If switching between synchronous operation and normal operation with signals, the same data need not be set for the master axis and slave axis, but data (1 to the number of controlled axes) must be set for both the master axis and slave axis.

1.6.2 Synchronization Establishment

Explanation

Upon power-up or after emergency stop cancellation, the machine positions on the master axis and slave axis under axis synchronous control are not always the same. In such a case, the synchronization establishment function matches the machine position on the master with that on the slave axis.

- Synchronization establishment method

Synchronization establishment based on machine coordinates

To perform synchronization establishment, set 1 in bit 7 (SOF) of parameter No. 8303 in order to enable synchronization establishment based on machine coordinates.

This method of synchronization establishment outputs the machine coordinate difference between the master axis and slave axis as command pulses for the slave axis to establish synchronization. A machine coordinate difference is output at a time as command pulses. So, if the compensation value is large, the machine abruptly makes a large movement. Taking this into consideration, set a maximum allowable compensation value to be used for synchronization establishment in parameter No. 8325. As a maximum allowable compensation value, set a maximum allowable value by which the machine may move abruptly. If a compensation value is larger than the value set in this parameter, an alarm (SV0001) is issued, and synchronization establishment is not performed. Moreover, when parameter No. 8325 is set to 0, synchronization establishment is not performed.

The result of comparing the positional difference between the master axis and slave axis with a maximum allowable compensation value for synchronization establishment can be checked using the synchronization establishment enable state output signal SYNOF <F0211>.

- First synchronization establishment after power-up

Two methods of performing the first synchronization establishment after power-up are available. One method is based on manual reference position return operation, and the other is based on absolute position detection.

In this case, a synchronization error check is made.

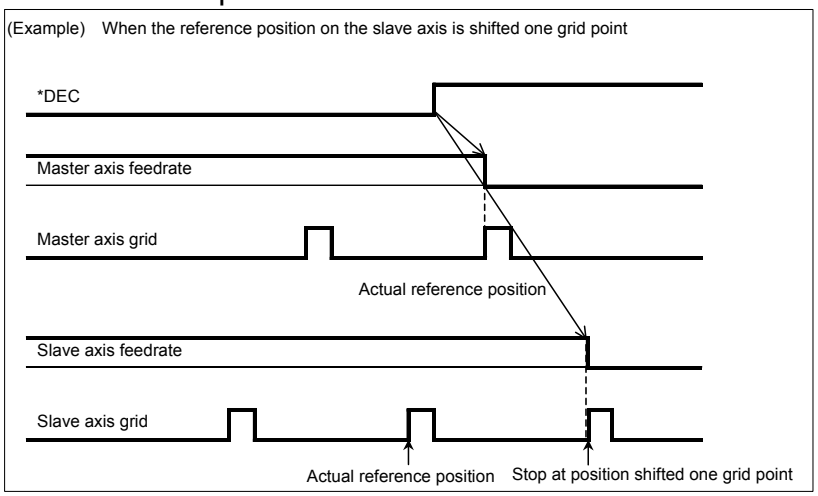
Synchronization establishment based on manual reference position return operation

When manual reference position return operation is performed along axes under axis synchronous control, the machine is placed at the reference position on the master axis and slave axis according to the same sequence as for normal reference position return operation.

The sequence is the same as the grid method for one axis only. However, only the deceleration signal for the master axis is used. When the deceleration signal is set to 0, the machine gradually stops along the master axis and slave axis, then an FL feedrate is set. When the deceleration signal is set to 1, the machine moves to a grid point along each of the master axis and slave axis, then stops.

NOTE

When the grid position difference between the master axis and slave axis is large, a reference position shift can occur, depending on the timing of the *DEC signal set to 1. In the example below, the shift along the slave axis is so large that the position shifted one grid point from the actual reference position is regarded as the reference position.



In such a case, match the grid position according to Subsection 1.6.4, "Automatic Setting for Grid Position Matching."

Synchronization establishment based on absolute position detection

When an absolute-position detector is used as the position detector, the machine positions on the master axis and slave axis are found at power-up time for automatic establish synchronization.

- Synchronization establishment after emergency stop cancellation, etc.

Synchronization establishment is also performed when servo position control is turned on, for example, at emergency stop cancellation, servo alarm cancellation, or servo-off cancellation time.
However, synchronization establishment is not performed at the time of axis removal cancellation. So, synchronization establishment based on manual reference position return operation is required as in the case of power-up time.

- One-direction synchronization establishment

Synchronization establishment can be performed by setting bit 0 (SSO) of parameter No. 8305 to 1 to move the machine in one direction along the master axis and slave axis. The move direction depends on the reference position setting based on bit 0 (SSA) of parameter No. 8304. When SSA = 0, for example, the machine coordinate on the master axis or slave axis, whichever larger, is used as the reference point. So, the machine moves in the + direction along the axes.
When bit 1 (SSE) of parameter No. 8305 is set to 1, normal synchronization establishment is performed instead of one-direction synchronization establishment after an emergency stop.

1.6.3 Automatic Setting for Grid Position Matching

Explanation

Before axis synchronous control can be performed, the reference position on the master axis must be matched with the reference position on the slave axis. With this function, the CNC automatically matches the reference positions (grid positions) on the master axis and slave axis under axis synchronous control.

[Operation procedure]

The procedure below is usable when bit 0 (ATE) of parameter No. 8303 is set to 1.

1. Set bit 1 (ATS) of parameter No. 8303 to 1.
2. Turn off the power then turn on the power.
3. Set the REF mode (or JOG mode in the case of reference position setting without dogs) when synchronous operation is ready, and make movements in the reference position return direction along the master axis and slave axis.
4. The movements along the master axis and slave axis automatically stop, and a grid difference value is set in parameter No. 8326. At this time, bit 1 (ATS) of parameter No. 8303 is set to 0, and the power-off request alarm (PW0000) is issued.
5. Turn off the power then turn on the power again.
6. Perform normal reference position return operation.

NOTE

1 Parameter setting
When bit 1 (ATS) of parameter No. 8303 is set, bit 4 (APZ) of parameter No. 1815 and parameter No. 8326 for the master axis and slave axis are set to 0. When the operator sets parameter No. 8326 (MDI, G10L52), bit 0 (ATE) of parameter No. 8303 is set to 0.

2 This function cannot be used together with the reference shift function.

1.6.4 Synchronization Error Check

Explanation

A synchronization error value is monitored at all times. If an error exceeding a certain limit is detected, an alarm is issued and the movement along the axis is stopped.
In this case, a synchronization error check based on machine coordinates is made.

- Synchronization error check based on machine coordinates

A synchronization error check based on machine coordinates is made.
The machine coordinate on the master axis is compared with that on the slave axis. When the error between the machine coordinates exceeds the value set in parameter No. 8314, the SV0005 alarm is issued, and the motor is stopped immediately.
This synchronization error check can also be made in the emergency stop, servo-off, and servo alarm state.
If a synchronization error check is made when the mode of operation is switched between synchronous operation and normal operation with an input signal, an error check is made even in normal operation. So, even if the axis synchronous control selection signal (SYNCx) or the axis synchronous control manual feed selection signal (SYNCJx) is set to 0 by mistake during synchronous operation, damage to the machine can be prevented.
The machine coordinates on the master axis and slave axis can be checked using the machine coordinate match state output signal SYNMT (Fn210).

- Synchronization error check based on a positional deviation value

The servo positional deviation value of the master axis and slave axis is monitored during axis synchronous control. When the positional deviation value exceeds the limit value set in parameter No. 8323, the DS0001 alarm is issued, and the axis synchronous control positional deviation error alarm signal (F403.0) is output.
The DS0001 alarm is issued to the master axis and slave axis.
When bit 4 (SYA) of parameter No. 8301 is set to 1, the positional deviation limit value of the master axis and slave axis is checked even if a servo-off occurs during axis synchronous control.

1.6.5 Methods of Alarm Recovery by Synchronization Error Check

Explanation

To recover from an alarm issued as a result of synchronization error check, two methods are available. One method uses the correction mode, and the other uses normal operation.
If the mode of operation is switched between synchronous operation and normal operation by using an input signal, only the method using normal operation can be used.
If synchronous operation is used at all times, only the method using the correction mode can be used.

- Procedure for correcting a synchronization error by using the correction mode

Use this method if synchronous operation is used at all times without using an input signal (when bit 5 (SCA) of parameter No. 8304 is set to 1).
When the correction mode is used, synchronization error check can be temporarily disabled, and a movement can be made along the master axis or slave axis to correct a synchronization error.
In the correction mode, synchronization compensation and error check are not performed, so that an alarm (DS0003) is issued as a warning.

1. Select the correction mode, and select an axis along which a movement is to be made by manual master axis feed. Set bit 2 (ADJ) of parameter No. 8304 of the master axis or slave axis to 1 to set the correction mode. Thus, by manual master axis feed, a movement can be made along the axis with this parameter set to 1.
   When this parameter is set to 1, the DS0003 (axis synchronous control correction mode) alarm is issued.
2. Reset the synchronization error excessive alarm.
   In this state, an error check is not made. Exercise due care.
3. Select the manual mode (jog, incremental feed, or handle).
4. While checking the synchronization error value, make a movement along the master axis or slave axis in the direction that reduces the error.
   If one master axis has multiple slave axes, an attempt to reduce the synchronization error of one slave axis by master axis movement may increase the synchronization error of another slave axis, thus disabling a movement in any direction. In such a case, by setting bit 4 (MVB) of parameter No. 8304 to 1, a movement can be made in a direction that increases the synchronization error.
5. When the synchronization error is reduced to within the allowable value for suppressing the alarm, reset the value of bit 2 (ADJ) of parameter No. 8304 to the original value to switch from the correction mode to the normal synchronization mode.
   Synchronization error check are restarted.
6. Reset the correction mode alarm.

- Method of recovery using normal operation

Use this method when switching between synchronous operation and normal operation by using an input signal.
Use the procedure below for recovery from alarm SV0005.

1. Set SYNCx/SYNCJx (with x representing a slave axis number) to 0 to select normal operation.
2. Set a value greater than the current value in the parameter No. 8314 for specifying a maximum allowable synchronization error, then reset the alarm.

3. Make a movement along the master axis or slave axis by using the manual handle so that the machine coordinates of the master axis and slave axis match to a maximum possible extent.
4. Return the value of parameter No. 8314 for specifying a maximum allowable synchronization error to the original value.

1.6.6 Axis Synchronous Control Torque Difference Alarm

Explanation

If a movement made along the master axis differs from a movement made along the slave axis during axis synchronous control, the machine can be damaged. To prevent such damage, the torque command difference between the two axes is observed. If the difference is abnormal, a servo alarm (SV0420) can be issued.

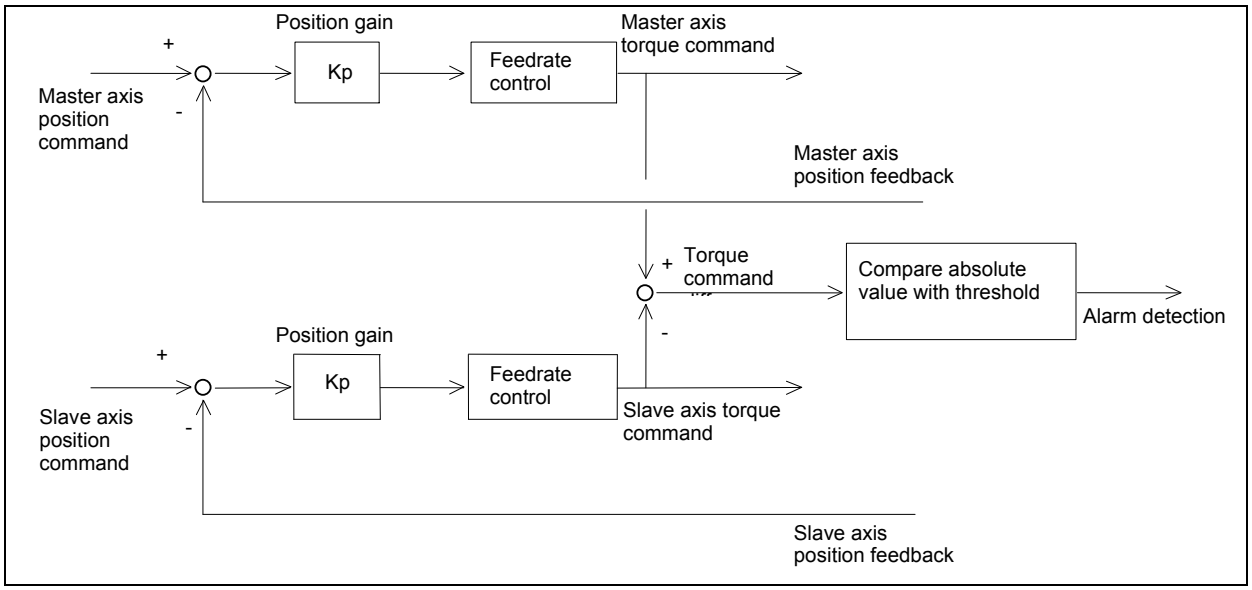


Fig. 1.6.6 (a) System configuration

[Method of use]

Specify the threshold parameter (No. 2031) according to the procedure below.

1. Set 0 in parameter No. 2031, and disable the torque difference alarm detection function.

2. To check the absolute value of the torque difference between the synchronous axes, set the parameters below. Set the same value for the two axes placed under axis synchronous control.
   Parameter No. 2115 = 0
   Parameter No. 2151 is as follows:
   - For a 2-path system (T series (2-path control), set 434 when the value of parameter No. 1023 is 1, 2, 5, 6, 9, or 10, or set 6578 when the value is 3, 4, 7, 8, 11, or 12.
   - For a one-path system, set 434.

3. Display the diagnostic screen by pressing the function key <SYSTEM> then the [DGNOS] soft key. Diagnose No. 0353 indicates the absolute value of the torque difference between the two axes.

4. Read the absolute torque difference value presented when normal operation is being performed. In the threshold parameter (No. 2031), set a value obtained by adding some margin to the read absolute value.
   An absolute torque difference value can be read using SERVO GUIDE Mate.

- Enabling/disabling of alarm detection

Alarm detection is enabled when the time set in parameter No. 8327 has elapsed after the servo ready signal SA <F000.6> is set to 1. When the input signal NSYNCA <G059.7> is set to 1, alarm detection is disabled.

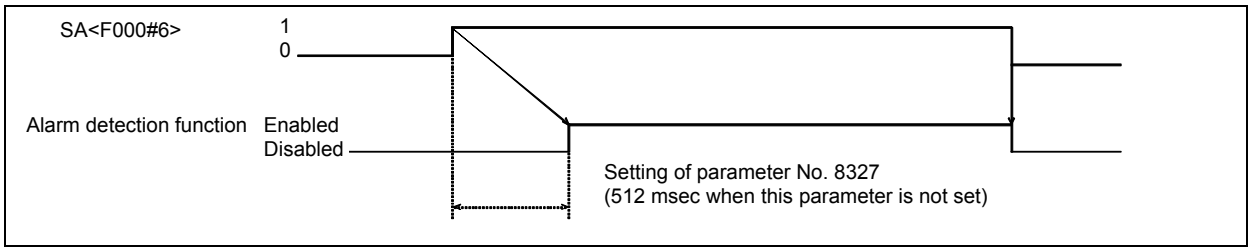


Fig. 1.6.6 (b) Timing chart

When the servo ready signal SA <F000.6>is set to 0, torque difference alarm detection is disabled.

NOTE
The servo axis number combination of the master axis and slave axis synchronized with each other must be such that an odd servo axis number is assigned to the master axis and the next servo axis number is assigned to the slave axis like (1,2) and (3,4).

1.6.7 Automatic Slave Axis Parameter Setting

Explanation

Axis synchronous control involoves parameters which must be set to the same values for the master axis and slave axis. (See the parameter list in "Parameters which must be set to the same value for the master axis and slave axis" to appear later. If bit 4 (SYPx) of parameter No. 8303 is 1, setting values for these parameters for the master axis causes the same values to be set for the parameters for the slave axis.
This function is enabled when parameters are set with the following methods:

- MDI key input
- Parameter file input
- Input with a program command
  (1) Programmable data input (parameter)
  (2) Stored stroke check 2 on (parameters Nos. 1322 and 1323)
  (3) Cutting condition selection function (parameters Nos. 1769, 13622, and 13623)
- Input with a signal
  Stroke limit external setting (M series) (parameters Nos. 1320 1321)
- Input with the FOCAS function and the window function

NOTE
Cutting condition selection function is optional fuction.

- Prohibition of writing for the slave axis

If bit 4 (SYPx) of parameter No. 8303 is 1, it is prohibited to set values to the parameters for the slave axis in "Parameters which must be set to the same value for the master axis and slave axis". Even if values are set, they will be invalid.
If any parameter other than the BIT parameter is set with the the methods below, either an alarm or a warning will be output.
MDI key input
A "WRITE PROTECT" warning is issued.
Programmable data input, stored stroke check 2 on
A "PS5379 WRITE PROTECTED TO SLAVE AXIS" alarm is issued.

Input with the FOCAS function and the window function
The return value is 7 (write protect error).

Signal

Signals for selecting the axis for axis synchronous control SYNC1 to SYNC5 <Gn138.0 to Gn138.4>

[Classification] Input signal

[Function] Axis synchronous control is performed during memory or MDI operation.
This signal is provided for each controlled axis. The number at the end of the signal name represents the number of the controlled axis.
SYNCx
x : 1 ..... The first axis becomes the slave axis for axis synchronous control.
2 ..... The second axis becomes the slave axis for axis synchronous control.
3 ..... The third axis becomes the slave axis for axis synchronous control.
: :

[Operation] When this signal is set to 1, the control unit operates as described below:
- During memory or MDI operation, the control unit issues the move command specified for the master axis to both the master axis and slave axis of axis synchronous control.

The master axis is specified with a parameter.

Signals for selecting the manual feed axis for axis synchronous control SYNCJ1 to SYNCJ5 <Gn140.0 to Gn140.4>

[Classification] Input signal

[Function] Axis synchronous control is performed in jog, handle, or incremental feed mode, or reference position return. This signal is provided for each controlled axis. The number at the end of the signal name represents the number of the controlled axis.
SYNCJx
x : 1 ..... The first axis becomes the slave axis for axis synchronous control.
2 ..... The second axis becomes the slave axis for axis synchronous control.
3 ..... The third axis becomes the slave axis for axis synchronous control.
: :

[Operation] When this signal is set to 1, the control unit operates as described below:
- In jog, handle, or incremental feed mode, the control unit issues the move command specified for the master axis to both the master axis and slave axis of axis synchronous control.

The master axis is specified with a parameter.

Signal for disabling torque difference alarm detection for axis synchronous control NSYNCA<Gn059.7>

[Classification] Input signal

[Function] When the torque difference alarm function for axis synchronous control is used, this signal can be used to disable alarm detection.

[Operation] When this signal is set to 1, torque difference alarm detection for axis synchronous control is disabled.

Machine coordinate match state output signals SYNMT1 to SYNMT5 <Fn210.0 to Fn210.4>

[Classification] Output signal

[Function] When master/slave axis pairs are set for axis synchronous control, this signal notifies an external unit that the machine coordinates of the master axis match those of the slave axis for each pair, regardless of the synchronous operation on or off state and servo ready state.

[Operation] When this signal is set to 1, the machine coordinates of the master axis match those of the slave axis. The signal corresponding to the pair of a master axis and the slave axis with the lowest axis number is output first and the machine coordinate status of up to five pairs can be checked.

Synchronization compensation enable state output signals SYNOF1 to SYNOF5 <Fn211.0 to Fn211.4>

[Classification] Output signal

[Function] When master/slave axis pairs are set for axis synchronous control, this signal notifies an external unit that the positional deviation difference between the master and slave axes is less than or equal to the maximum compensation for synchronization for each pair, regardless of the synchronous operation on or off state and servo ready state.

[Operation] When this signal is set to 1, the positional deviation difference between the master and slave axes is less than or equal to the maximum compensation for synchronization. The signal corresponding to the pair of a master axis and the slave axis with the lowest axis number is output first and whether to enable synchronization compensation can be checked for up to five pairs. This signal is not output for each axis.

Signal for indicating a positional deviation error alarm for axis synchronous control SYNER<Fn403.0>

[Classification] Output signal

[Function] When the positional deviation check function is used for axis synchronous control, this signal notifies an external unit that the alarm is issued.

[Operation] When axis synchronous control is applied, the servo positional deviation of the master axis and that of the slave axis are monitored. If the limit set in parameter No. 8323 is exceeded, alarm DS0001 is issued and the signal for indicating a positional deviation error alarm for axis synchronous control (F403.0) is set to 1.
This signal is set to 0 when the alarm is cleared by a reset. This signal is not output for each signal.

Axis synchronous control status signals SYNO1 to SYNOF5 <Fn211.0 to Fn211.4>

[Classification] Output signal

[Function] These signals notify that axis synchronous control is in progress.

[Operation] These signals become 1 in the following case:
- When the corresponding axis is in axis synchronous control.

They become 0 in the following case:
- When the corresponding axis is not in the axis synchronous control.

NOTE
Whether axis synchronous control is in progress does not necessarily correspond to individual selection signals/parameters (feed axis synchroniztion control selection signal, axis synchronous control manual feed selection signal, bit 5 (SCA) of parameter No. 8304). This signal is 0 in the emergency stop, servo alarm, serve off, and axis detach states.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn059	NSYNCA							
Gn138				SYNC5	SYNC4	SYNC3	SYNC2	SYNC1
Gn140				SYNCJ5	SYNCJ4	SYNCJ3	SYNCJ2	SYNCJ1
Fn210				SYNMT5	SYNMT4	SYNMT3	SYNMT2	SYNMT1
Fn211				SYNOF5	SYNOF4	SYNOF3	SYNOF2	SYNOF1
Fn403								SYNER
Fn532				SYNO5	SYNO4	SYNO3	SYNO2	SYNO1

- How signals for each axis are handled

Some signals for each axis are input or output to both the master and slave axes and others are input or output to only the master axis. The following table lists the type of each signal.

- Input signals

Address	Bit	Symbol	Signal name	Master and slave axes	Master axis only
Gn100	0	+Jx	Feed axis direction select signals		O
Gn102	0	-Jx	Feed axis direction select signals		O
Gn104	0	+EXLx	Stored stroke limit 1 switching signals in axis direction	O	
Gn105	0	-EXLx	Stored stroke limit 1 switching signals in axis direction	O	
Gn106	0	MIx	Mirror image signals		O
Gn108	0	MLKx	Machine lock signal in each axis		O
Gn110	0	+LMx	Stroke limit external setting signals	O	
Gn112	0	-LMx	Stroke limit external setting signals	O	
Gn114	0	*+Lx	Over travel signals		O
Gn116	0	*-Lx	Over travel signals		O
Gn118	0	*+Edx	External deceleration signals		O
Gn120	0	*-Edx	External deceleration signals		O
Gn124	0	DTCHx	Controlled axes detach signals	O *1	
Gn126	0	*SVFx	Servo-off signals	O *1	
Gn130	0	*ITx	Interlock signals in each axis		O
Gn138	0	SYNCx	Signals for selecting the axis for axis synchronous control	O	
Gn140	0	SYNCJx	Signals for selecting the manual feed axis for axis synchronous control	O	
Gn192	0	IGVRYx	VRDY off alarm ignore signal in each axis	O *1	
X009	0	*DECx	Deceleration signals for reference position return		O

- Output signals

Address	Bit	Symbol	Signal name	Master and slave axes	Master axis only
Fn094	0	ZPx	Reference position return completion signals	O	
Fn096	0	ZP2x	2nd reference position return completion signals	O	
Fn098	0	ZP3x	3rd reference position return completion signals	O	
Fn100	0	ZP4x	4th reference position return completion signals	O	
Fn102	0	MVx	Axis moving signals	O	
Fn104	0	INPx	In-position signals	O	
Fn106	0	MVDx	Axis moving direction signals	O	
Fn108	0	MMIx	Mirror image check signals	O	
Fn110	0	MDTCHx	Controlled axis detach status signals	O	
Fn120	0	ZRFx	Reference position establishment signals	O	

NOTE
1 Turn the signal marked with *1 on or off simultaneously for the master and slave axes in the synchronization mode.
2 In the above table, the address only for the 1st axis is listed. For the addresses of the 2nd and subsequent axes, see "SIGNAL SUMMARY" in "Connection Manual."

Parameter

2031	Torque-command-difference threshold for a torque-difference alarm

[Input type] Parameter input
[Data type] Word axis
[Min. unit of data] 0 to 14564

An alarm is issued when the absolute value of a torque-command-difference between two axes exceeds this value.
Set the same value for the two axes under axis synchronous control.
In addition, for the master and slave axes synchronized with each other, the master axis must be indicated by an odd number and the slave axis must be indicated by the next even number such as (1, 2) or (3, 4) in specifying the combination of the servo axes.

	#7	#6	#5	#4	#3	#2	#1	#0
8301				SYA				

[Input type] Parameter input
[Data type] Bit path

#4 SYA In the servo-off state in axis synchronous control, the limit of the difference between the positioning deviation of the master axis and that of the slave axis is:
0: Checked.
1: Not checked.

	#7	#6	#5	#4	#3	#2	#1	#0
8302	SMA							

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#7 SMA When an absolute position detector is attached, and bit 4 (APZ) of parameter No. 1815 for an axis in synchronous operation is set to OFF, APZ of the pairing axis in synchronous operation is:
0: Not set to OFF.
1: Set to OFF.

	#7	#6	#5	#4	#3	#2	#1	#0
8303	SOFx			SYPx		SAFx	ATSx	ATEx

[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 ATEx In axis synchronous control, automatic setting for grid positioning is:
0: Disabled
1: Enabled
Set this parameter with a slave axis.

#1 ATSx In axis synchronous control, automatic setting for grid positioning is:
0: Not started
1: Started
Set this parameter with a slave axis.

NOTE
When starting automatic setting for grid positioning, set parameter ATS to 1. Upon the completion of setting, parameter ATS is automatically set to 0.

#2 SAFx In axis synchronous control, a movement along a slave axis is:
0: Not added to actual feedrate display.
1: Added to actual feedrate display.
Set this parameter with a slave axis.

#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 The parameters that are automatically set are found in "Parameters which must be set to the same value for the master and slave axes" to appear later.
2 Set this parameter for both the master and slave axes.

#7 SOFx In axis synchronous control, the synchronization establishment function based on machine coordinates is:
0: Disabled.
1: Enabled.
Set this parameter with a slave axis.

	#7	#6	#5	#4	#3	#2	#1	#0
8304	SYEx		SCAx	MVBx		ADJx		SSAx

[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 SSAx When the one-direction synchronization establishment function under axis synchronous control is used:
0: The axis with a larger machine coordinate is used as the reference.
1: The axis with a smaller machine coordinate is used as the reference.

> **NOTE**
> Set this parameter (SSA) to the same value for both the master and slave axes.

#2 ADJx In axis synchronous control, this parameter specifies an axis along which a movement is made in the modification mode.
0: A movement is not made in the modification mode along the axis.
1: A movement is made in the modification mode along the axis.
When this parameter is set to 1, the modification mode is set.
Along an axis with this parameter set to 1, a movement is made by a move command for the master axis.
Set this parameter for one of the master and slave axes.
When there are multiple slave axes for one master axis, set this parameter to 1 for an axis with which a synchronization error excessive alarm is issued for recovery. If an alarm is issued with multiple axes, modify this parameter after recovery of one axis to recover another axis.

#4 MVBx In the modification mode, a move command in a direction that increases a synchronization error is:
0: Ignored.
1: Valid.
When there are multiple slave axes for one master axis, an attempt to reduce the synchronous error of a slave axis by a movement along the master axis can increase the synchronization error of another slave axis. If this parameter is set to 0 in such a case, a movement can be made in neither direction along the master axis. In this case, set bit 2 (ADJ) of parameter No. 8304 to make a movement along a slave axis to perform a corrective operation.

#5 SCAx In axis synchronous control:
0: Synchronous operation is performed when the axis synchronous control manual feed selection signal SYNCJ or the axis synchronous control selection signal SYNC for slave axes is set to 1.
1: Synchronous operation is performed at all times.
Set this parameter with a slave axis.

#7 SYEx When external machine coordinate system shift is specified by external data input/output for the master axis in synchronous control, the slave axis is:
0: Not shifted.
1: Shifted by the same amount as specified for the master axis.
Set this parameter with a slave axis.
This function is disabled during normal operation.

	#7	#6	#5	#4	#3	#2	#1	#0
8305						SRF	SSE	SSO

[Input type] Parameter input
[Data type] Bit path

#0 SSO The uni-directional synchronization function in axis synchronous control is:
0: Disabled.
1: Enabled.

#1 SSE After emergency stop, the uni-directional synchronization function in axis synchronous control is:
0: Disabled.
1: Enabled.

#2 SRF In axis synchronous control, G27, 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.

8311	Axis number of master axis in axis synchronous control

NOTE
Set this parameter to the same value for both the master and slave axes.

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to Number of controlled axes

Select a master axis in axis synchronous control. In the parameter for the slave axis, set the axis number of the master axis.

[Example 1]

When one set of axis synchronous control is used:
When the master axis is the first axis (X-axis), and the slave axis is the third axis (Z-axis), set parameter No. 8311 as follows:
Parameter No.8311 X (first axis) = 0
Parameter No.8311 Y (second axis) = 0
Parameter No.8311 Z (third axis) = 1
Parameter No.8311 A (fourth axis) = 0

[Example 2]

When two sets of axis synchronous control is used:
When the master axes are the first axis and second axis, and the slave axes are the fourth axis and third axis, set parameter No. 8311 as follows:
Parameter No.8311 X (first axis) = 0
Parameter No.8311 Y (second axis) = 0
Parameter No.8311 Z (third axis) = 2
Parameter No.8311 A (fourth axis) = 1

8312	Enabling/disabling mirror image in axis synchronous control

[Input type] Parameter input
[Data type] Word axis
[Valid data range] -127 to 128

This parameter sets mirror image for the slave axis. When 100 or a more value is set with this parameter, the mirror image function is applied to synchronous control. Set this parameter to the slave axis.

[Example]

For reverse synchronization with the master axis being the third axis and the slave axis being the fourth axis, set parameter No. 8312 as follows:
Parameter No.8312 X (first axis) = 0

Parameter No.8312 Y (second axis) = 0
Parameter No.8312 Z (third axis) = 0
Parameter No.8312 A (fourth axis) = 100

NOTE
In synchronous operation with mirror image applied, synchronization establishment, synchronization error checking, and modification mode cannot be used.

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).
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) 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) is issued, and the synchronization establishment is not performed.

Specify a slave axis for this parameter. To enable this parameter, set the parameter SOF (bit 7 of parameter No.8303) to 1. When 0 is set in this parameter, synchronization establishment is not performed.

8326	Difference between master axis and slave axis reference counters

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

The difference between the master axis reference counter and slave axis reference counter (master axis and slave axis grid shift) is automatically set when automatic setting for grid positioning is performed. Then, the difference is transferred together with an ordinary grid shift value to the servo system when the power is turned on. This parameter is set with a slave axis.

8327	Torque difference alarm detection timer

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] msec
[Valid data range] 0 to 4000

This parameter sets a time from the servo preparation completion signal, SA <F000.6>, being set to 1 until torque difference alarm detection is started in axis synchronous control.
When 0 is set in this parameter, the specification of 512 msec is assumed.
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 99999999

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 99999999

This parameter specifies an M code for switching from normal operation to synchronous operation.
The M code set in this parameter is not buffered.

Notes on setting parameters for each axis

Parameters to be set for each axis can be divided into the following four types when they are set for an axis under axis synchronous control:

(1) Parameter which must be set to the same value for the master and slave axes
(2) Parameter which needs to be set only for the master axis (data for the slave axis is not used.)
(3) Parameter which may be set to different values for the master and slave axes
(4) Parameter which needs to be set only for the slave axis (data for the master axis is not used.)

Parameters are listed for each type below. If a parameter is not listed in any table below, assume that the parameter is of type (1) and set the same value for the master and slave axes.

⚠ CAUTION
1 If different values are set in a parameter of type (1) for the master and slave axes, these axes may not operate as axes under axis synchronous control.
2 When a signal is used to switch between synchronous and normal operation, in a parameter of type (2), also set a value for each of the master and slave axes.

(1) Parameters which must be set to the same value for the master and slave axes

Parameter number	Description
12#7	Releasing the assignment of the controlled axis for each axis
1005#0	Whether reference position return has been performed
1005#1	Enabling setting the reference position without dogs
1005#4	Enabling the external deceleration signal for the positive direction in cutting feed for each axis
1005#5	Enabling the external deceleration signal for the negative direction in cutting feed for each axis
1005#6	Turning off only motor activtion in axis detach
1005#7	Enabling the controlled axis detach signal for each axis
1006#0,#1	Setting a rotary axis
1006#3	Specifying move commands for each axis using diameter programming
1006#5	Direction of manual reference position return
1007#1	Using the same method as for manual reference position return to perform automatic reference position return (G28)
1007#3	The rotary axis command in an absolute command is the absolute position of the specified value for the end point coordinates and the sign of the specified value for the direction of rotation.
1007#4	Method of setting the reference position without dogs
1008#0	Enabling rotation axis roll-over
1008#1	Not making a short cut in an absolute command
1008#2	Rounding relative coordinates with the travel distance of one rotation.
1240	Coordinate value 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
1260	Amount of a shift per one rotation of a rotation axis
1310#0,#1	Enabling stored stroke check 2, 3
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
1322	Coordinate value I of stored stroke check 2 in the positive direction on each axis
1323	Coordinate value I of stored stroke check 3 in the negative direction on each axis
1324	Coordinate value I of stored stroke check 3 in the positive direction on each axis
1325	Coordinate value I of stored stroke check 2 in the negative direction on each axis
1326	Coordinate value II of stored stroke check 1 in the negative direction on each axis
1327	Coordinate value II of stored stroke check 1 in the negative direction on each axis
1420	Rapid traverse rate for each axis
1610#0,#1	Acceleration type of cutting feed and dry run
1610#4	Acceleration type of jog feed
1620	Time constant used for linear acceleration/deceleration for each axis
1621	Time constant T2 used for bell-shaped acceleration/deceleration in rapid traverse for each axis
1622	Time constant of acceleration/deceleration in cutting feed for each axis
1623	FL rate of acceleration/deceleration in cutting feed for each axis
1624	Time constant of acceleration/deceleration in jog feed for each axis.
1625	FL rate of exponential acceleration/deceleration in jog feed for each axis
1626	Time constant of exponential acceleration/deceleration during thread cutting cycle
1627	FL rate of acceleration/deceleration during thread cutting cycle

Parameter number	Description
1671	Maximum permissible acceleration of acceleration/deceleration before interpolation for linear rapid traverse for each axis
1763	FL rate acceleration/deceleration after cutting feed interpolation for each axis in the mode of acceleration/deceleration before interpolation for each axis
1769	Time constant of acceleration/deceleration after cutting feed interpolation in the mode of acceleration/deceleration before interpolation for each axis
1815#2	Using a linear scale having reference marks
1818#0,#1,#3, 1819#2	Related to the linear scale with absolute addressing reference marks/linear scale with an absolute addressing origin
1819#0	Follow-up in the servo off state
1819#1	Reference position establishment state at the time of a servo alarm
1819#7	Enabling advance feed forward
1821	Reference counter capacity
1825	Servo loop gain
1881	Group number when an abnormal load is detected
1882	Intervals of mark 2 of the linear scale with absolute addressing reference marks.
1885	Maximum allowable value for total travel during torque control
1886	Positional deviation when torque control is canceled
2028	Limit speed for enabling position gain switching
2031	Torque-command-difference threshold for a torque-difference alarm
2060	Torque limit
2068	Feed forward coefficient
2092	Advance feed forward coefficient
2144	Position feed forward coefficient for cutting
2178	Position gain for rapid traverse
2179	Reference counter (denominator)
3605#0	Using bidirectional pitch error compensation
3624	Interval between pitch error compensation positions for each axis
3625	Travel distance per revolution in pitch error compensation of rotation axis type
8304#0	Setting the reference position for the unidirectional synchronization compensation function
7310	Order of axes to move to with dry run after a program restart (in the case of not switching between sychrnonous operation and normal operation with signals)
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 0)

(2) Parameters which need to be set only for the master axis

Parameter number	Description
1005#3	Operation to be performed during a manual reference position return when a reference position is established
1012#0	Method of setting the reference position without dogs
1250	Coordinates of the reference position when automatic coordinate system setting is performed
1408#0	The rotary axis feedrate control method is to convert the rotation speed on the circumference of a virtual circle.
1421	F0 rate of rapid traverse override for each axis
1423	Feedrate in manual continuous feed (jog feed) for each axis
1424	Manual rapid traverse rate for each axis
1425	FL rate of the reference position return for each axis
1427	External deceleration rate of rapid traverse for each axis
1428	Reference position return speed for each axis
1430	Maximum cutting feedrate for each axis

Parameter number	Description
1432	Maximum cutting feedrate for each axis in the mode of acceleration/deceleration before interpolation
1465	Radius of the virtual circle in a rotary axis virtual circle velocity command
1660	Maximum permissible acceleration of acceleration/deceleration before interpolation for each axis
1671	Maximum permissible acceleration of acceleration/deceleration before interpolation for linear rapid traverse for each axis
1735	Permissible acceleration for each axis in the deceleration function with the acceleration in circular interpolation
1737	Permissible acceleration for each axis in the deceleration function with the acceleration in AI contour control
1783	Permissible speed difference in speed determination with the speed difference at a corner
1788	Permissible acceleration change amount in speed determination with the acceleration change on each axis
1789	Permissible acceleration change amount in speed determination with the accleration change on each axis (linear interpolation)
3455#0	When the decimal point is omitted, mm, inch, sec units (pocket calculator type decimal point programming)
5401#0	Enabling scaling (M series)
5022	Distance (L) from reference tool tip position to the reference measurement surface
5421	Scaling magnification (M series) for each axis
5440	Positioning direction and overrun distance in single directional positioning
7010	Axis that can be specified with a manual numeric command
7181	First retract distance at butt-type reference position setting
7182	Second retract distance at butt-type reference position setting
7183	First retract speed at butt-type reference position setting
7184	Second retract speed at butt-type reference position setting
7185	Retract speed at butt-type reference position setting (common to the first and second retractions)
7186	Torque limit value at butt-type reference position setting
7741	Retracted distance
8410	Allowable feedrate difference used for feedrate determination, based on a corner feedrate difference

(3) Parameters which may be set to different values for the master and slave axes

Parameter number	Description
1020	Program axis name for each axis
1023	Number of the servo axis for each axis
18XX 2XXX	Parameters related to servo (other than those listed in (1), (2), or (4))
1936	Connector number of the first separate detector interface unit
1937	Connector number of the second separate detector interface unit
1938	Connector number of the third pulse module
1939	Connector number of the fourth pulse module
3115#0	Not displaying the current position
3115#1	Not displaying the absolute and relative coordinates
3115#3	Not adding axis moving to actual cutting feedrate display
3131	Subscript of axis name of synchronous controlled axis and tandem control axis
3620	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
3622	Number of the pitch error compensation position at extremely positive position for each axis
3623	Magnification for pitch error compensation for each axis
3626	Number of the both-direction pitch error compensation position at extremely negative position (for movement in the negative direction)
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

Parameter number	Description
5861 to 5864	Inclination compensation : Compensation point number for each axis
5871 to 5874	Inclination compensation : Compensation α at compensation point number for each axis
7310	Order of axes to move to with dry run after a program restart (in the case of switching between sychrnonous operation and normal operation with signals (SYNC1 to SYNC5<Gn138> and SYNCJ1 to SYNCJ5<Gn140>))

(4) Parameters which need to be set only for the slave axis

Parameter number	Description
1817#6	Performing tandem control for the axis
8303#0	Enabling automatic setting of grid positioning
8303#1	Setting the start of automatic setting of grid positioning
8303#2	Adding data for the slave axis for actual cutting feedrate display
8303#7	Using the synchronization compensation function based on machine coordinates for the axis synchronous control function
8304#5	Always performing axis synchronous control
8304#7	Enabling external machine coordinate system shift
8311	Axis number of master axis in axis synchronous control
8312	Enabling/disabling mirror image in axis synchronous control
8314	Maximum allowable error in synchronization error check based on machine coordinates
8323	Limit in positional deviation check in axis synchronous control
8325	Maximum compensation value in synchronization establishment based on machine coordinates
8326	Difference between master axis and slave axis reference counters
8327	Torque difference alarm detection timer

Diagnosis

The synchronization error and compensation are displayed on the diagnostic screen.

3500	Synchronization error for each axis

[Unit of data] Detection unit

[Description] The difference between the positions of the master and slave axes (synchronization error) is displayed.
It is displayed for the axis number of the slave axis.

Alarm and message

Number	Message	Description
PS0213	ILLEGAL COMMAND IN SYNCHRO-MODE	In axis synchronous 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 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.
PS5379	WRITE PROTECTED TO SLAVE AXIS	It is not possible to directly set the parameters for the slave axis under axis synchronous control.
DS0001	SYNC EXCESS ERROR (POS DEV)	In axis synchronous 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 only for the slave axis.
DS0003	SYNCHRONIZE ADJUST MODE	The system is in the synchronize adjust mode.

Number	Message	Description
SV0001	SYNC ALIGNMENT ERROR	In axis synchronous control, the amount of compensation for synchronization exceeded the parameter (No. 8325) setting value. This alarm occurs only for a slave axis.
SV0005	SYNC EXCESS ERROR (MCN)	In axis synchronous 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.
SV0420	SYNC TORQUE EXCESS	In axis synchronous 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.

Caution

⚠ CAUTION

1 When making a synchronization error check, ensure that the reference position on the master axis and the reference position on the slave axis must be at the same position.
2 In manual reference position return operation, the same operation is performed along the master axis and slave axis until a deceleration operation starts. After a deceleration operation starts, grid detection is performed for the master axis and slave axis independently of each other.
3 Pitch error compensation and backlash compensation are performed for the master axis and slave axis independently of each other.

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 7 (SRF) of parameter No. 8304.
   <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 During synchronous operation, the axis-by-axis signals such as for external deceleration, interlock, and machine lock are enabled for the master axis only. During synchronous operation, those signals for the slave axis are ignored.
4 When switching the synchronization state in a program, be sure to specify M codes (parameter No. 8337 and No. 8338) for turning synchronization on and off. By switching between the input signals SYNCx (G138) and SYNCJx (G140) from the PMC with the M codes, the synchronization state can be switched in the program.

NOTE

5 When controlled axis removal is performed, the synchronization state is cancelled. When performing controlled axis removal, perform removal for the master axis and slave axis at the same time.
6 If a programmed command is specified for the slave axis during synchronous operation, an alarm (PS0213) is issued.
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 (SCA) of parameter No. 8304 set to 0) to select normal operation.
7 Axis synchronous control and PMC axis control cannot be used at the same time.
8 When bit 4 (SYPx) of parameter No. 8303 is changed from 0 to 1 in automatic slave axis parameter setting, those slave axis parameters that have already been set are not automatically set. Automatic slave axis parameter setting is enabled after parameter No. 8311 and bit 4 (SYPx) of parameter No. 8303 are set.
9 If inputting a parameter file after enabling automatic slave axis parameter setting, set parameter No. 8311 and bit 4 (SYPx) of parameter No. 8303 for both the CNC and the parameter file before inputting it.

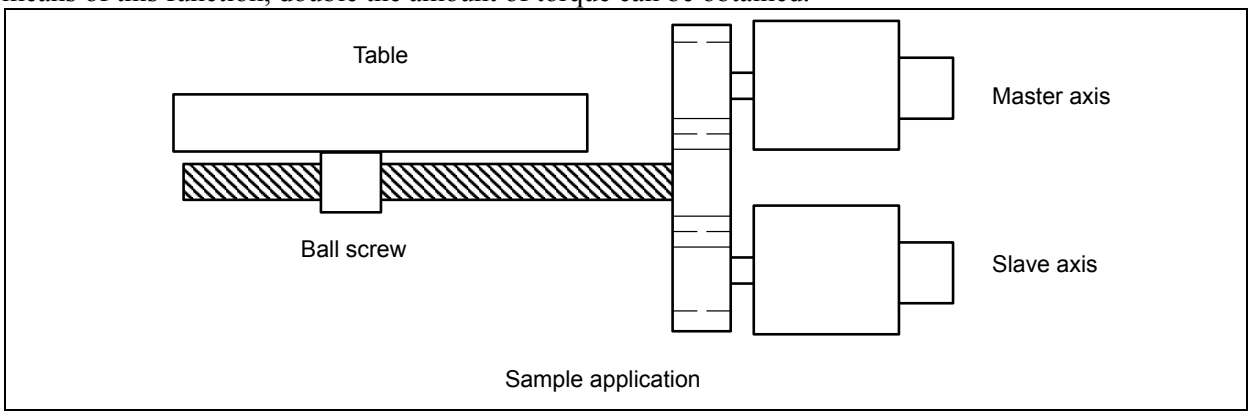
Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Axis synchronous control

1.7 TANDEM CONTROL

Overview

If a single motor cannot produce sufficient torque to move a large table, for example, this function allows two motors to be used. By means of this function, two motors can be used to perform movement along a single axis.
Positioning is carried out only for the master axis. The slave axis is used only to produce a torque. By means of this function, double the amount of torque can be obtained.



Sample application

The CNC generally processes the two axes of tandem control as a single axis. In the management of servo parameters and the monitoring of servo alarms, however, the two axes are handled individually.

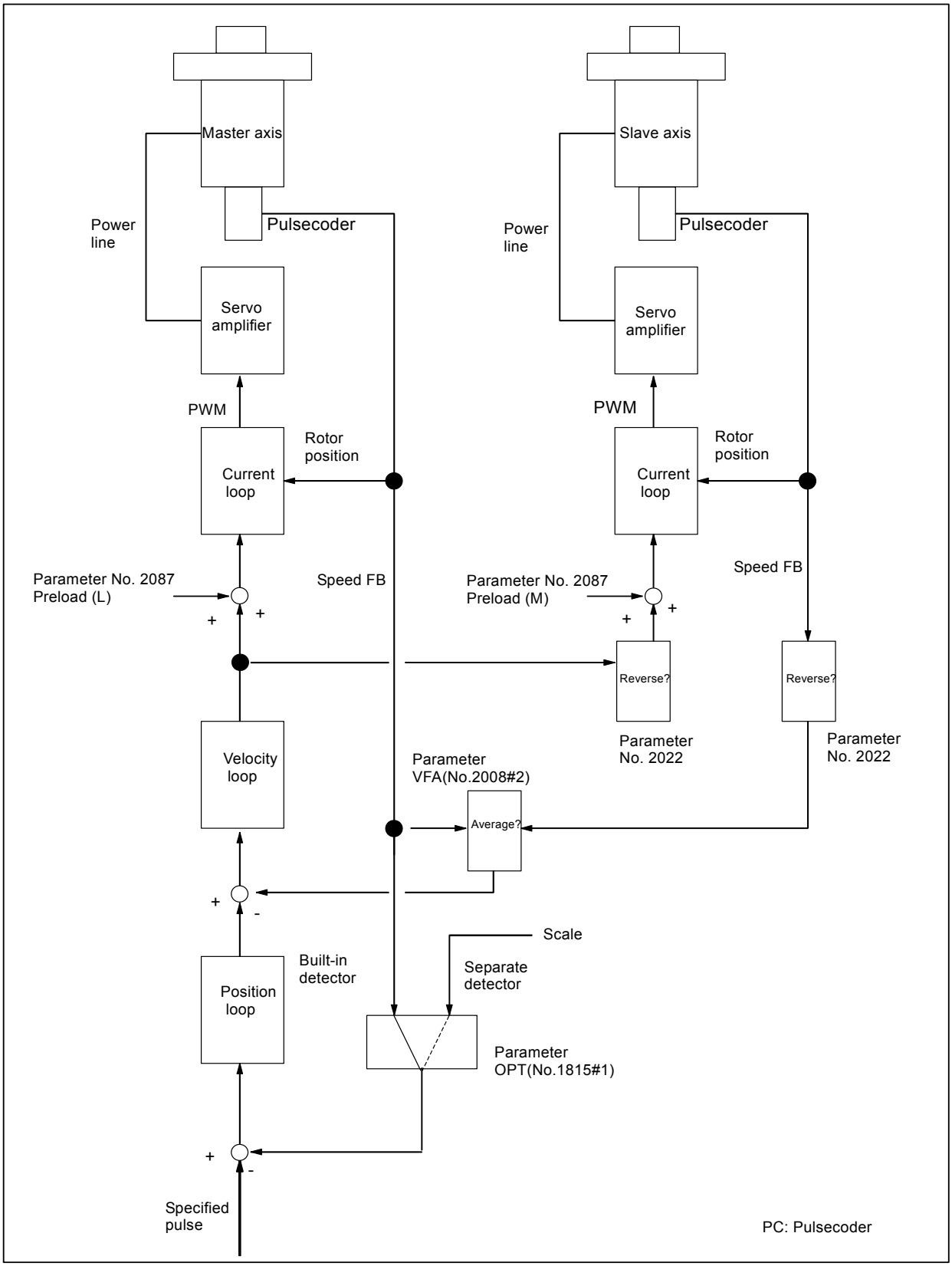


Fig. 1.7 (a) Block Diagram of Tandem Control

Explanation

- Axis configuration in tandem control

To specify the axis configuration in tandem control, follow the procedure below:

(1) Tandem control can be performed for up to four pairs of axes.
It can be performed for up to twelve pairs of axes for each path.

(2) In terms of controlled axes, the pair of axes is handled as two separate axes. For a programmed command or manual feed, the pair of axes is handled as a single axis.

(3) The pair of axes is handled as two separate axes in the management of servo parameters and the monitoring of servo alarms.

(4) Assign two consecutive numbers, that is one odd and one even number, to the master and slave axes as their servo axis numbers (parameter 1023). Assign the smaller number to the master axis.
(Example)
If the servo axis number of the master axis (parameter 1023) is set to 1, specify servo axis number 2 for the corresponding slave axis. If the servo axis number of the master axis is set to 3, specify servo axis number 4 for the corresponding slave axis.

(5) The master and slave axes may have the same name or different names.

(6) A subscript can be attached to an axis name like X1, X2, XM, and XS. If the same axis name is used for multiple axes, and a unique subscript is assigned to each of those axes, the axes can be distinguished from each other on the screen display, or which of those axes issued an alarm can be identified.
Set a subscript in parameter No. 3131.

(7) The slave axis is handled as a controlled axis. Set bit 0 (NDPx) of parameter No. 3115 to 1 to suppress the position display.

The following sample axis configuration is for a machine with five axes X, Y, Z, A, and B. The X-axis and Y-axis are the master axes of tandem control.
Number of controlled axes = Seven

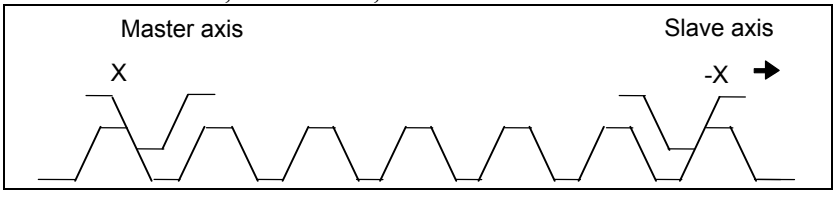
Axis number	Displayed axis name	Axis name (No.1020)	Subscript (No.3131)	Servo axis number (No.1023)	Tandem axis (No.1817#6)	
1	XM	88	77	1	1	Master axis of tandem control
2	XS	88	83	2	1	Slave axis of tandem control
3	Z	90	0	5	0	
4	A	65	0	6	0	
5	B	66	0	7	0	
6	YM	89	77	3	1	Master axis of tandem control
7	YS	89	83	4	1	Slave axis of tandem control

(8) The master and slave axes must be included in the same path.

(9) Set an absolute position detector only on the master axis. If it is set on the slave axis, alarm SV0006 "ILLEGAL TANDEM AXIS" is issued.

- Preload function

By adding an offset to the torque controlled by the position (velocity) feedback device, the function can apply opposite torques to the master and slave axes so that equal and opposite movements are performed for both axes. This function can reduce the effect of backlash on the master and slave axes caused by the tandem connection of the two motors via a gear. This function, however, cannot reduce backlash between the ball screw and table or other backlash inherent to the machine.
If a preload of x is set for the master axis and -x for the slave axis, the opposing preload torques are continuously applied to the two axes, even at rest, as shown below:



⚠ CAUTION
1 Specify as low a preload as possible. Avoid specifying a preload higher than the rated torque. Too high a preload will trigger an overload alarm because the specified torques continue to be applied, even at rest. A preload that is only slightly higher than the frictional force is recommended. Thus, the recommended preload may be about one-third of the rated torque.
2 If the motors rotate in opposite directions (different signs are specified in parameter 2022), specify the preload values with the same sign.

- Velocity feedback average function

As shown in the block diagram of tandem control, the motor of the slave axis is not subject to velocity control. A machine with a large amount of backlash may become unstable if the motor of the slave axis vibrates as a result of backlash in the gear. This can be overcome by applying velocity control to the slave axis also. This velocity feedback average function is enabled when bit 2 of parameter 2008 is set to 1.

- Improved stability of a closed-loop system

The following two functions can increase the stability and position gain of a closed-loop system having a linear scale:
- Dual position feedback function
- Machine velocity feedback function

For details of these functions, refer to FFANUC AC SERVO MOTOR α*is*/α*i* series PARAMETER MANUAL (B-65270EN).

- Notes on stability of tandem control

An important factor affecting stability in tandem control is the capability of back feed. Back feed is to cause movement along either the master or slave axis from the other axis, via the transmission mechanism connecting the two axes. A machine without this capability may be inclined to become unstable and require adjustments.

- Connection of axis signals

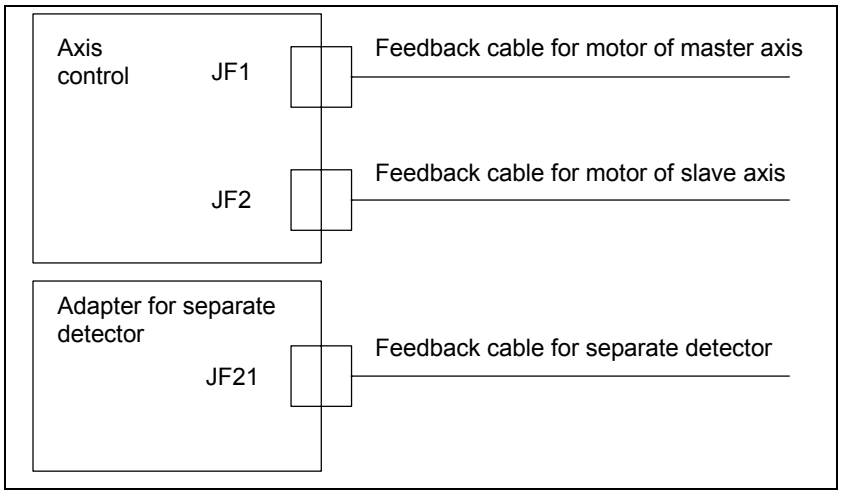
The DI/DO signals, generally connected to each axis, must be connected only to the master axis of two axes of tandem control. The signals need not be connected to the slave axis. The following signals, however, may have to be connected depending on the application.

i) Controlled axis detach signal and servo off signal
Connect these signals so that the master and slave axis signals are simultaneously input.
ii) Overtravel limit signal
Connect the signal so that 1 is always output as the overtravel limit signal for the slave axis.

If the slave axis stroke limit must also be detected, connect the signals so that the signal detected on the slave axis is sent to the overtravel limit signal of the master axis.

- Connecting motors

Connect the motors according to the servo axis numbers. Connect the feedback cable of the slave axis.
(Sample connection for position feedback cable)



- Servo alarms

Motor overload and other servo alarms are displayed separately for the master and slave axes.

Parameter

- Setting data (parameters)

The parameters that are generally set for each axis can, when set for axes under tandem control, be classified into the following three groups:

i) Parameters in which identical values must be set for the master and slave axes
ii) Parameters that must be specified only for the master axis (The corresponding parameter for the slave axis is not used.)
iii) Parameters for which different values may be set for the master and slave axes

The classifications of the parameters are described below. Any parameter that is not listed in the tables for the three classifications should be processed as a parameter of type i) and, specify identical values for the master and slave axes.

> ⚠ **CAUTION**
> Note that, if different values are set for the master and slave axes in a parameter of type i), the operations for the two axes of tandem control will not be performed correctly.

- Care must be taken to specify the following two servo parameters, according to the directions of rotation around the master and slave axes.
  No. 2022 Direction of rotation of the motor
  No. 2087 Preload value
  In parameter 2022, specify 111 for forward rotation and -111 for the reverse rotation.
  In parameter 2087, specify values having identical signs when the motors of the master and slave axes rotate in opposite directions. Specify values having different signs when the motors of the master and slave axes rotate in the same direction.
- If a separate pulse coder is used, use of the separate pulse coder must be set for the master axis. For the slave axis, use of a built-in pulse coder must be set. Therefore, pay particular attention to setting the following parameters.

No. 1815#1	Separate pulse coder
No. 1816#6 to #4	Detection multiplier (DMR)
No. 2024	Number of position detection feedback pulses (PPLS)
No. 1821	Capacity of an optional reference counter
No. 2084	Numerator of flexible feed gear ratio
No. 2085	Denominator of flexible feed gear ratio

If, for example, a motor with serial pulse coder A is used with a linear scale capable of detecting a position in 1-μm units, and if a single rotation of the motor produces a movement of 4 mm, specify the parameters as shown below:

		Master axis	Slave axis
No. 1815#1	=	1	0
No. 1816	=	01110000	01110000
No. 2024	=	4000	12500
No. 1821	=	4000	4000
No. 2084	=	0	4
No. 2085	=	0	1000

- Parameters that should be set only for the master axes

Parameter No.	Meaning of parameters
0012#0	Mirror image
0012#7	Servo control off
1004#7	Input unit 10 times
1005#4	External deceleration in plus direction
1005#5	External deceleration in minus direction
1005#7	Servo control off
1022	Parallel axis specification
1220	External workpiece coordinate shift
1221	Workpiece zero point offset by G54
1222	Workpiece zero point offset by G55
1223	Workpiece zero point offset by G56
1224	Workpiece zero point offset by G57
1225	Workpiece zero point offset by G58
1226	Workpiece zero point offset by G59
1423	Jog feedrate
1424	Manual rapid traverse
1425	FL rate in manual reference position return
1427	External deceleration rate at rapid traverse
1430	Maximum cutting feedrate
1815#1	Separate type pulse coder
1815#5	Absolute pulse coder
2008#2	Velocity feedback average function

- Parameters that may be set to different values for the master and slave axes

Parameter No.	Meaning of parameters
1020	Axis name
1023	Servo axis number
2022	Motor rotation direction
2087	Preload value
3115	Current position display
1310#0	Soft OT2
1310#1	Soft OT3
1320	1st stroke limit I of plus side
1321	1st stroke limit I of minus side
1322	2nd stroke limit of plus side
1323	2nd stroke limit of minus side
1324	3rd stroke limit of plus side
1325	3rd stroke limit of minus side
1326	1st stroke limit II of plus side
1327	1st stroke limit II of minus side
1815#1	Separate type pulse coder
1816#6 to #4	Detection multiplier (DMR)
1821	Arbitrary reference counter capacity
2024	Position detection feedback pulses (PPLS)

Parameter No.	Meaning of parameters
2084	Numerator of flexible feed gear ratio
2085	Denominator of flexible feed gear ratio

- Parameters that should be set to the same values for the master and slave axes

Parameter No.	Meaning of parameters
1006#7	Least command increment (0.0001mm)
1005#0	Whether reference position return has been performed
1005#1	Enabling setting the reference position without dogs
1006#0	Rotary axis
1006#1	Machine coordinate of rotary axis is rotary type
1006#3	Diameter/radius specification
1006#5	Direction of reference position return
1240	Reference position as viewed from machine zero
1241	Coordinate of 2nd reference position
1242	Coordinate of 3rd reference position
1243	Coordinate of 4th reference position
1260	Move distance per rotation of rotary axis
1420	Rapid traverse rate
1421	F0 of rapid traverse override
1620	Time constant of rapid traverse linear acceleration/deceleration
1621	Time constant of rapid traverse bell shaped acceleration/deceleration
1622	Time constant of exponential acceleration/deceleration in cutting feed
1623	FL of exponential acceleration/deceleration in cutting feed
1624	Time constant exponential acceleration/deceleration in jog feed
1625	FL of exponential acceleration/deceleration in jog feed
1626	Time constant of exponential acceleration/deceleration during thread cutting cycle
1627	FL of exponential acceleration/deceleration during thread cutting cycle
1820	Command multiplier (CMR)
18XX	Most of parameters related to digital servo
20XX	Most of parameters related to digital servo

	#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
2008						VFA		

[Input type] Parameter input
[Data type] Bit axis

#2 VFA In tandem control, the speed feedback average function is:
0: Disabled.
1: Enabled.

2087	Preload value (Tcmd offset)

[Input type] Parameter input
[Data type] Word axis
[Unit of data] (Amplifier limit) / 7282
[Valid data range] -1821 to 1821
This parameter prevents backlash by adding an offset to the torque command.
Set a value slightly greater than the friction value.
The estimated value is approximately 1/3 of the rated torque.

(Example) When a torque equivalent to 3 A is set in two opposite directions, if the amplifier limit is 40 A
3 / (40 / 7282) = 546
Master side = 546
Slave side = -546

2021	Load inertia ratio

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 32767
(Load inertia)/(Motor inertia) x 256

For tandem control,
(Load inertia)/(Motor inertia) x 256/2
Set the same value for the master and slave axes.

2022	Motor rotation 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] -111 to 111
This parameter sets the direction in which the motor rotates.
To set the clockwise direction as viewed from the pulse coder, set 111. To set the counterclockwise direction as viewed from the pulse coder, set -111.
When the master axis and slave axis rotate in opposite directions each other, this parameter is used for switching.

Alarm and message

Number	Message	Description
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).
SV1055	ILLEGAL TANDEM AXIS	The setting of parameter No. 1023 is invalid for tandem control.
SV1056	ILLEGAL TANDEM PAIR	The setting of parameter bit 6 (TDM) of parameter No. 1817 is invalid for tandem control.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Tandem control

1.8 ARBITRARY ANGULAR AXIS CONTROL

Overview

When the angular axis installed makes an angle other than 90° with the perpendicular axis, the arbitrary angular axis control function controls the distance traveled along each axis according to the inclination angle as in the case where the angular axis makes 90° with the perpendicular axis.
Arbitrary axes can be specified as a set of an angular axis and perpendicular axis by parameter setting.
The actual distance traveled is controlled according to an inclination angle. However, a program, when created, assumes that the angular axis and perpendicular axis intersect at right angles. The coordinate system used at this time is referred to as the program coordinate system. (The program coordinate system may be referred to as the Cartesian coordinate system, and the actual move coordinate system may be referred to as the slanted coordinate system or machine coordinate system.)

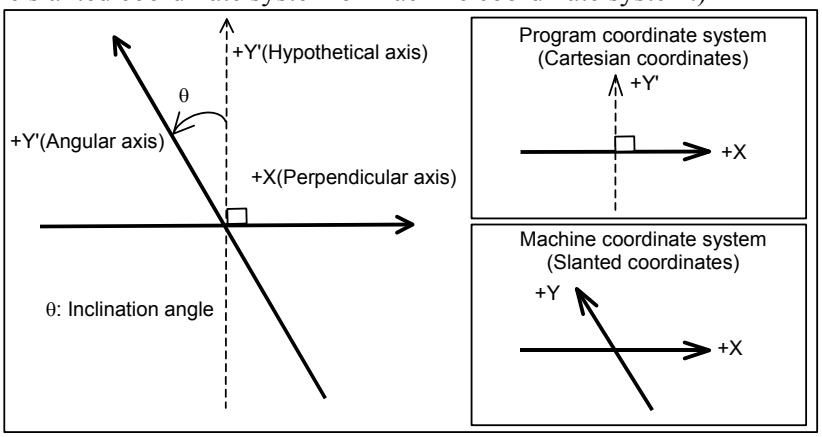


Fig. 1.8 (a)

NOTE
Arbitrary angular axis control is optional function.

Explanation

When the amounts of travel along the angular axis and the perpendicular axis are Ya and Xa, respectively, the amounts are controlled according to the formulas shown below.

$$Ya = \frac{Yp}{\cos\theta}$$

Xa,*Ya*: Actual distance
Xp,*Yp*: Programmed distance

The amount of travel along the perpendicular axis is corrected by the influence of travel along the angular axis, and is determined by the following formula:

$$Xa = Xp - C \times Yp \times \tan\theta$$

NOTE
The coefficient C is 1/2 in the case of diameter specification for the perpendicular axis (X) or 1 in the case of radius specification.

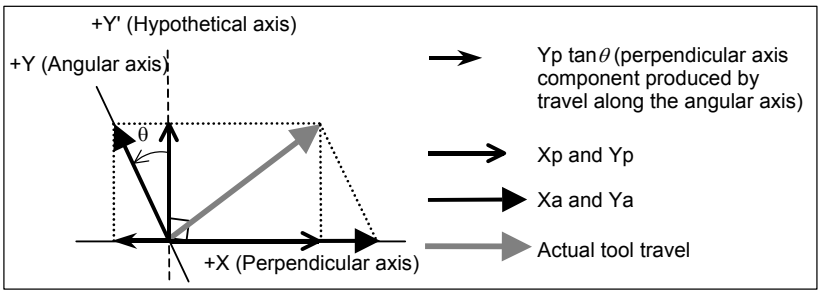


Fig. 1.8 (b)

- Feedrate

When the Y-axis is an angular axis, and the X-axis is a perpendicular axis, the feedrate along each axis is controlled as described below so that the feedrate in the tangent direction becomes Fp.
The feedrate component along the Y-axis is determined by the following expressions:

$$Fay = \frac{Fp}{\cos\theta}$$

Fa represents the actual feedrate.
Fp represents a programmed feedrate.

$$Fax = Fp - Fp \times \tan\theta$$

- Absolute and relative position display

An absolute and a relative position are indicated in the programmed Cartesian coordinate system.

- Machine position display

A machine position indication is provided in the machine coordinate system where an actual movement is taking place according to an inclination angle.

Method of use

The angular and perpendicular axes for which arbitrary angular axis control is to be applied must be specified beforehand, using parameters (Nos. 8211 and 8212). When 0 is set in one of the parameters, the same number is specified in the parameters, or a number other than the controlled axis numbers is specified in a parameter, however, an angular axis and perpendicular axis are selected according to the table below.

	Angular axis	Perpendicular axis
M series	Y-axis of the basic three axes (axis with 2 set in parameter No. 1022)	Z-axis of the basic three axes (axis with 3 set in parameter No. 1022)
T series	X-axis of the basic three axes (axis with 1 set in parameter No. 1022)	Z-axis of the basic three axes (axis with 3 set in parameter No. 1022)

- Parameter AAC (No. 8200#0) enables or disables the arbitrary angular axis control. If the function is enabled, the distance traveled along each axis is controlled according to an angular angle parameter (No. 8210).
- By using bit 2 (AZR) of parameter No. 8200, whether to make a movement along the perpendicular axis by a movement made along the angular axis when a manual reference position return operation is performed along the angular axis can be chosen. When a movement along the perpendicular axis is enabled (AZR = 1), a reference position return operation along the perpendicular axis can be performed by a movement made along the angular axis.
- By setting the normal axis/arbitrary angular axis control invalid signal NOZAGC <Gn063.5> to 1, slanted axis control only for the angular axis can be available. In this time the angular axis are converted to those along the slanted coordinate system without affecting commands to normal axis. Use this signal when operating each axis independently.

- Manual reference position return operation

A movement is made to the reference position (machine position) set in parameter No. 1240. By using bit 2 (AZR) of parameter No. 8200, whether to make a movement along the perpendicular axis when a reference position return operation is performed along the angular axis can be chosen.

- Automatic reference position return operation (G28, G30)

A movement to the middle point along the angular axis affects a movement along the perpendicular axis. It is possible to select between Cartesian coordinate system operation (compatible with FS0*i*) and angular coordinate system operation as the movement along the angular axis from the intermediate position to the reference position, using bit 0 (ARF) of parameter No. 8209.

If manual reference position return operation is not performed even once after the power is turned on, operation is performed in the same sequence as for manual reference position return operation. So, specify commands first for the angular axis then for the perpendicular axis.

Example 1)

When the Y-axis is an angular axis and the X-axis is a perpendicular axis

(1) If the angular axis is first specified then the perpendicular axis is specified, reference position return operation is performed normally.

```
G28Y_;
G28X_;
```

(2) If the perpendicular axis is first specified then the angular axis is specified, or if the perpendicular axis and the angular axis are specified at the same time, alarm PS0372 is issued when a movement is made along the perpendicular axis.

```
{G28X_;  or  {G28X_Y_;
{            {
{G28Y_;      {
```

Example 2)

Automatic reference position return example

(If the Y-axis is an angular axis, the X-axis is a perpendicular axis, and the angular angle is -30°)

<1> Automatic reference position return command on the Y-axis from point P2
>G91G28X200.

<2> Automatic reference position return command on the X-axis from point P1
>G91G28Y100.

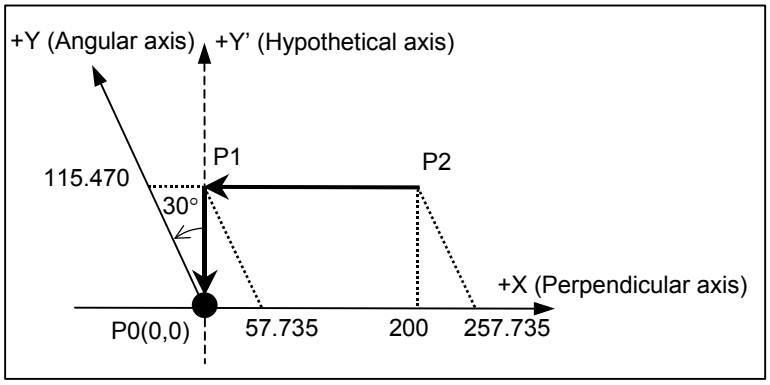
(1) If bit 0 (ARF) of parameter No. 8209 is 1 (compatible with FS0*i*)

<1> Coordinates at P1

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	57.735
Y	100.000	Y	115.470

<2> Coordinates at P0

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	0.000
Y	0.000	Y	0.000



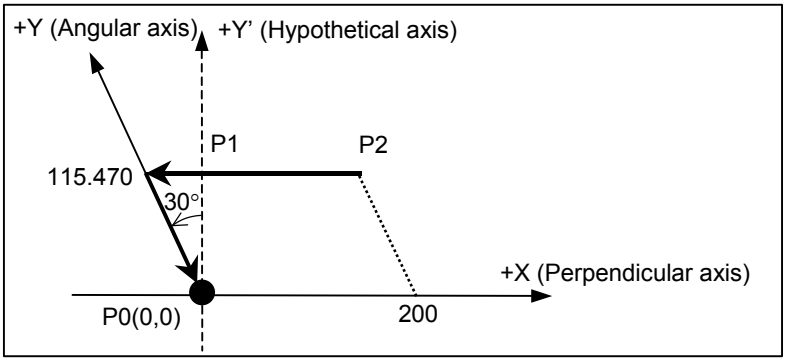
(2) If bit 0 (ARF) of parameter No. 8209 is 0

<1> Coordinates at P1

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	0.000
Y	100.000	Y	115.470

<2> Coordinates at P0

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	0.000
Y	0.000	Y	0.000



- Machine coordinate selection (G53)

By specifying (G90)G53X_Y_: (when the Y-axis is an angular axis, the X-axis is a perpendicular axis, and the inclination angle is -30°), a movement is made by rapid traverse.

However, a movement along the angular axis (G53 command) does not affect a movement along the perpendicular axis, regardless of whether the perpendicular axis/arbitrary angular axis control disable signal (NOZAGC) is turned on or off.

Example)

<1> Move command for movement from point P0 to point P1

```
>G90G53Y100.
```

<2> Move command for movement from point P1 to point P2

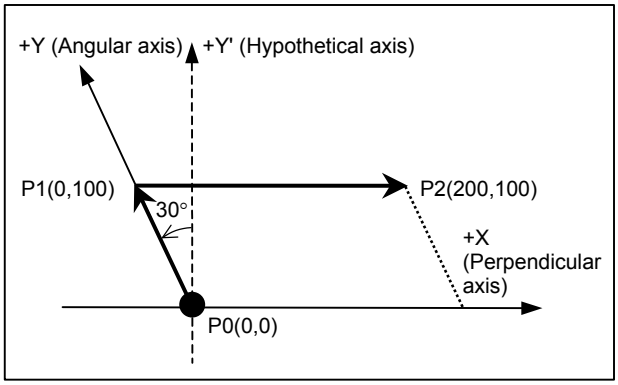
```
>G90G53X200.
```

<1> Coordinates of P1

(Absolute coordinate)		(Machine coordinate)	
X	-50.000	X	0.000
Y	86.603	Y	100.000

<2> Coordinates of P2

(Absolute coordinate)		(Machine coordinate)	
X	150.000	X	200.000
Y	86.603	Y	100.000



- Commands for linear interpolation and positioning of linear interpolation type (G01, G00)

The tool moves to a specified position in the Cartesian coordinate system when the following is specified:
(G90)G00X_Y_; (when the Y-axis is an angular axis, the X-axis is a perpendicular axis, and the inclination angle is -30°)
or
(G90)G01X_Y_F_; (when the Y-axis is an angular axis, the X-axis is a perpendicular axis, and the inclination angle is -30°)

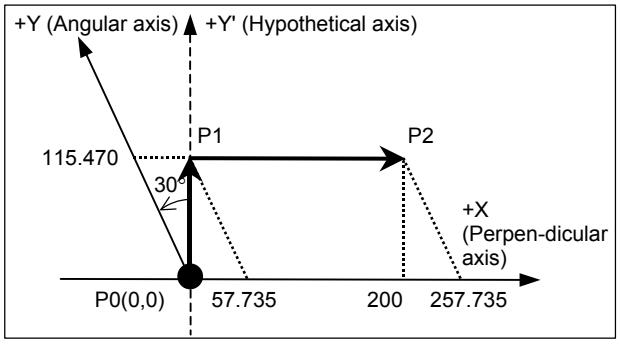
Example) Examples of positioning
<1> Move command for movement from point P0 to point P1
>G90G00Y100.
<2> Move command for movement from P1 to P2
>G90G00X200.

(1) When the perpendicular axis/arbitrary angular axis control disable signal (NOZAGC) is set to 0
<1> Coordinates of P1

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	57.735
Y	100.000	Y	115.470

<2> Coordinates of P2

(Absolute coordinate)		(Machine coordinate)	
X	200.000	X	257.735
Y	100.000	Y	115.470

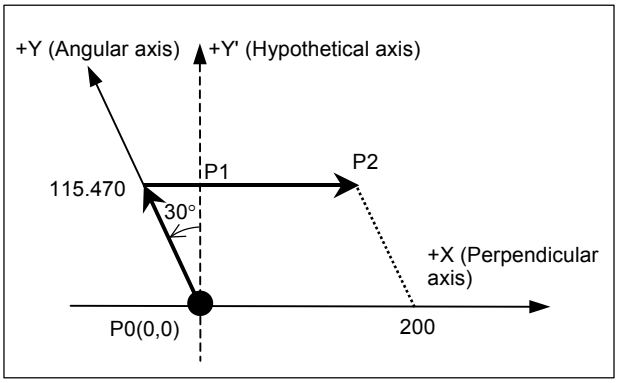


(2) When the perpendicular axis/arbitrary angular axis control disable signal (NOZAGC) is set to 1
<1> Coordinates of P1

(Absolute coordinate)		(Machine coordinate)	
X	0.000	X	0.000
Y	100.000	Y	115.470

<2> Coordinates of P2

(Absolute coordinate)		(Machine coordinate)	
X	200.000	X	200.000
Y	100.000	Y	115.470



- Stored stroke limit

Stored stroke limits under arbitrary angular axis control can be set not in a slanted coordinate system but in the Cartesian coordinate system by setting bits 0, 1, and 2 (AOT, AO2, and AO3) of parameter No. 8201.

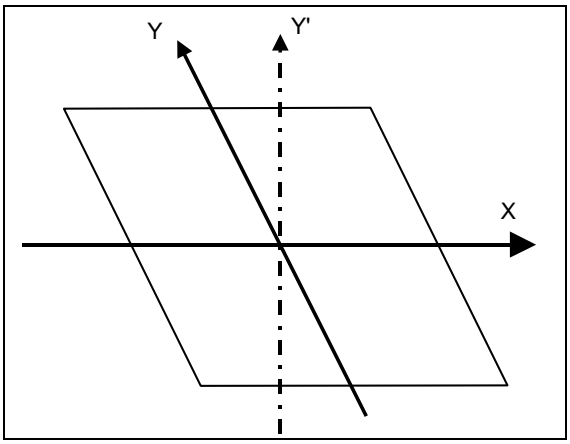


Fig. 1.8 (c) OT area in a slanted coordinate system

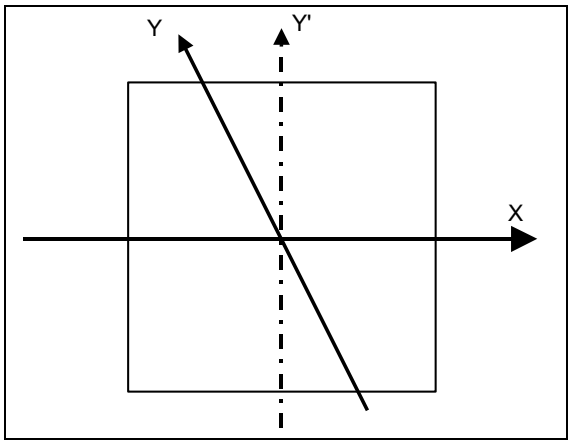


Fig. 1.8 (d) OT area in a Cartesian coordinate system

Machine coordinates include a value converted for the angular axis and a compensation value for the perpendicular axis, so that a slanted machine coordinate system as shown in Fig. 1.8 (c) results.
A stored stroke limit is checked in the machine coordinate system, so that the limit area is slanted to form a rhombus as shown in Fig. 1.8 (c). In this case, the area cannot be identified intuitively. So, stroke limits are checked not in an actual slanted machine coordinate system but in a virtual Cartesian machine coordinate system as shown in Fig. 1.8 (d).

The functions that operate in the Cartesian coordinate system are:

- Stored stroke limit 1 (Both of I and II)
- Stored stroke limit 2 (G22/G23)
- Stored stroke limit 3
- Pre-movement stroke check

The pre-movement stroke check function does not work in a slanted coordinate system. Unless this function is enabled, and the coordinate system is converted to the Cartesian coordinate system, no stroke check is made.

- Stored stroke external setting (function specific to the M series only and valid only for OT1)
- Bit 7 (BFA) of parameter No. 1300 for specifying whether to issue an alarm before or after a stroke limit is exceeded

The stored stroke limit functions other than the above work in a slanted coordinate system.

- Relationships between this function and axis-by-axis input/output signals

The table below indicates the relationships between this function and the meaning of each controlled axis signal.
The input/output signals are classified as signals valid for the program coordinate system (Cartesian coordinate system) and signals valid for the machine coordinate system (slanted coordinate system). In the "Classification" column, "Cartesian" is indicated for a signal that is valid for the Cartesian coordinate system, and "Slanted" is indicated for a signal that is valid for the slanted coordinate system.
A signal valid for the Cartesian coordinate system means a signal valid for a specified axis, and a signal valid for the slanted coordinate system is a signal valid for actual machine movement.
This means that when a movement is made along the perpendicular axis by a movement along the angular axis alone:

A signal valid for the Cartesian coordinate system is affected by a movement along the angular axis.
A signal valid for the slanted coordinate system is not affected by a movement along the angular axis.

Input signal				
Signal name		**Address**	**Classification**	**Remarks**
Axis-by-axis interlock	*ITx	G130	Cartesian	When a movement is made along the angular axis only, interlocking the perpendicular axis does not interlock a movement along the perpendicular axis made by a movement along the angular axis. Caution) When using the axis-by-axis interlock signal, make both of the angular axis and perpendicular axis high.
Overtravel	*+Lx *-Lx	G114 G116	Slanted	This signal is applied to each axis independently. (If the perpendicular axis is made high, no alarm is issued for the perpendicular axis even when an OT alarm is issued for the angular axis.)
Deceleration signal for reference position return	*DECx	X009	Slanted	This signal is applied to each axis independently.
Servo-off signal	SVFx	G126	Slanted	This signal is applied to each axis independently.
Controlled axis removal signal	DTCHx	G124	Slanted	This signal is applied to each axis independently.
Feed axis direction selection signal	+Jx -Jx	G100 G102	Cartesian	A movement is made in the Cartesian coordinate system. (When the +J/-J signal for the angular axis is made high, a movement is made also along the perpendicular axis.)
Mirror image	MIx	G106	Slanted	Mirror image is applied to the slanted coordinate system for each axis independently. Caution) Be sure to turn off the mirror image signal for the angular axis and perpendicular axis engaged in manual operation.
Axis-by-axis machine lock	MLKx	G108	Slanted	This signal is applied to each axis independently.

Output signal				
Signal name		**Address**	**Classification**	**Remarks**
In-position signal	INPx	F104	Slanted	Applied to each axis independently.
Mirror image check signal	MMIx	F108	Slanted	Applied to each axis independently.
Controlled axis removal in-progress signal	MDTCHx	F110	Slanted	Applied to each axis independently.
Travel in-progress signal	MVx	F102	Slanted	Applied to each axis independently.
Reference position return completion signal	ZPx	F094	Cartesian	Applied to each axis independently. (A manual reference position return operation and the first automatic reference position return operation after power-up need to be performed first for the angular axis.)
2nd reference position return completion signal	ZP2x	F096	Cartesian	Applied to each axis independently.
3rd reference position return completion signal	ZP3x	F098	Cartesian	Applied to each axis independently.

Output signal				
Signal name		Address	Classification	Remarks
4th reference position return completion signal	ZP4x	F100	Cartesian	Applied to each axis independently.

Limitation

- Linear scale with an absolute address reference mark

- For both of the angular axis and perpendicular axis, a linear scale with an absolute address reference mark must be used.
- Reference position return operation must be first completed along the angular axis.
- Return operation cannot be performed along the perpendicular axis while return operation is being performed along the angular axis.

- Synchronous control (T series (2-path control))

- To perform synchronous control over related axes under arbitrary angular axis control, specify the angular axis and Cartesian axis on the master axis side and the angular axis and Cartesian axis on the slave axis side as the targets of synchronous control at the same time. In synchronous control, one angular axis must be paired with the other angular axis while one Cartesian axis must be paired with the other Cartesian axis.
  If synchronous control is performed in a way other than the above, alarm PS0375 is issued.
  Example)

Path 1		Path 2
X1 (Cartesian axis)	←Synchronous control pair→	X2 (Cartesian axis)
Y1 (slanted axis)	←Synchronous control pair→	Y2 (slanted axis)

- Composite control (T series (2-path control))

- To perform composite control over related axes under arbitrary angular axis control, specify the angular axis and Cartesian axis on the master axis side and the angular axis and Cartesian axis on the slave axis side as the targets of composite control at the same time. In composite control, one angular axis must be paired with the other angular axis while one Cartesian axis must be paired with the other Cartesian axis.
  If composite control is performed in a way other than the above, alarm PS0375 occurs.
  Example)

Path 1		Path 2
X1 (Cartesian axis)	←Composite control pair→	X2 (Cartesian axis)
Y1 (slanted axis)	←Composite control pair→	Y2 (slanted axis)

- Rigid tapping

- An angular axis cannot be used as a tapping axis for rigid tapping.

- Functions that cannot be used simultaneously

- Axis synchronous control, polygonal turning (T series), hypothetical axis control, hobbing machine function, EGB function (M series), PMC axis control, superimposed control (T series (2-path control))

Signal

Signal for disabling arbitrary angular axis control for the perpendicular axis NOZAGC<Gn063.5>

[Classification] Input signal

[Function] Disables arbitrary angular axis control for the perpendicular axis.

[Operation] When this signal is set to 1, the control unit operates as follows:
- Converts an angular axis move command to angular coordinates. The perpendicular axis is not affected by an angular axis move command, however.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8200						AZR		AAC

[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 AAC
0: Does not perform arbitrary angular axis control.
1: Performs inclined axis control.

#2 AZR
0: The machine tool is moved along the Cartesian axis during manual reference position return along the slanted axis under arbitrary angular axis control.
1: The machine tool is not moved along the Cartesian axis during manual reference position return along the slanted axis under arbitrary angular axis control.

	#7	#6	#5	#4	#3	#2	#1	#0
8201	ADG					AO3	AO2	AOT

[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 AOT Stored stroke limit 1 under arbitrary angular axis control is handled as:
0: Value in the slanted coordinate system.
1: Value in the Cartesian coordinate system.

#1 AO2 Stored stroke limit 2 under arbitrary angular axis control is handled as:
0: Value in the slanted coordinate system.
1: Value in the Cartesian coordinate system.

#2 AO3 Stored stroke limit 3 under arbitrary angular axis control is handled as:
0: Value in the slanted coordinate system.
1: Value in the Cartesian coordinate system.

#7 ADG The contents of diagnostic data Nos. 306 and 307 are:
0: Not swapped. The slanted axis and Cartesian axis are displayed in this order.
1: Swapped. The Cartesian axis and slanted axis are displayed in this order.

⚠ CAUTION
1 After arbitrary angular axis control parameter setting, be sure to perform manual reference position return operation.

⚠ CAUTION

2 Before manual reference position return operation is performed along the perpendicular axis, reference position return operation along the angular axis must be completed (with the reference position return completion signal for the angular axis (ZPx) set to 1). If reference position return operation is performed along the perpendicular axis first, an alarm (PS0372) is issued.
3 When the setting is made so that the tool moves along the perpendicular axis during manual reference position return along the slanted axis (bit 2 (AZK) of parameter No. 8200 is set to 0), if once manual reference position return has been performed along the angular axis, also perform manual reference position return along the perpendicular axis immediately after the operation.
4 Before attempting to manually move the tool along the angular and perpendicular axes simultaneously, set perpendicular/arbitrary angular axis control disable signal NOZAGC to 1.
5 Once the tool has been moved along the angular axis when perpendicular/arbitrary angular axis control disable signal NOZAGC has been set to 1, manual reference position return must be performed.
6 The same increment system must be used with the angular axis and perpendicular axis.
7 Before a perpendicular axis reference position return check can be made, angular axis reference position return operation must be completed.
8 No rotary axis must be set for the angular axis and perpendicular axis. A rotary axis may be specified only for a linear axis.
9 Set a position switch operation range (parameter Nos. 6930 to 6965) in a slanted coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
8209								ARF

[Input type] Parameter input
[Data type] Bit path

NOTE

When this parameter bit is set, the power must be turned off before operation is continued.

#0 ARF In arbitrary angular axis control, a movement from an intermediate point to the reference position in the G28/G30 command is:
0: Made in the slanted coordinate system.
1: Made in the Cartesian coordinate system.

8210	Slant angle of a slanted axis in arbitrary angular axis control

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the applied axis

[Valid data range] -180.000 to 180.000. However, arbitrary angular axis control is disabled in the ranges -95.000 to -85.000 and 85.000 to 95.000 (in the case of IS-B).

8211	Axis number of a slanted axis subject to arbitrary angular axis control

8212	Axis number of a Cartesian axis subject to slanted axis control

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] 1 to number of controlled axes

When arbitrary angular axis control is to be applied to an arbitrary axis, these parameters set the axis numbers of a slanted axis and Cartesian axis. If 0 is set in either of the two parameters, the same number is set in the two parameters, or a number other than the controlled axis numbers is set in either of the two parameters, a slanted axis and Cartesian axis are selected as indicated in the following table:

	Slanted axis	Cartesian axis
M series	Y-axis (axis with 2 set in parameter No. 1022) of the basic three axes	Z-axis (axis with 3 set in parameter No. 1022) of the basic three axes
T series	X-axis (axis with 1 set in parameter No. 1022) of the basic three axes	Z-axis (axis with 3 set in parameter No. 1022) of the basic three axes

Alarm and message

Number	Message	Description
PS0372	REFERENCE RETURN INCOMPLETE	An attempt was made to perform an automatic return to the reference position on the orthogonal 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 arbitrary 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 orthogonal axis.
PS0375	CAN NOT ANGULAR CONTROL(SYNC:MIX:OVL)	Arbitrary angular axis control is disabled for this axis configuration. 1) When some related axes under arbitrary angular axis control are not in synchronous control mode or when one angular axis is not paired with the other angular axis or one Cartesian axis is not paired with the other Cartesian axis in synchronous control 2) When some related axes under composite control are not in composite control mode or when one angular axis is not paired with the other angular axis or one Cartesian axis is not paired with the other Cartesian axis in composite control 3) When related axes under arbitrary angular axis control is switched to superimposed control mode1)

Diagnosis

306	Machine coordinates on the angular axis in the Cartesian coordinate system

307	Machine coordinates on the perpendicular axis in the Cartesian coordinate system

[Data type] Real number

[Unit of data] Machine unit
Machine coordinates in the Cartesian coordinate system are displayed in arbitrary angular axis control.
Bit 7 (ADG) of parameter No. 8201 can be used to change the display order.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Arbitrary angular axis control

1.9 ELECTRONIC GEAR BOX (M SERIES)

1.9.1 Electronic Gear Box (M Series)

M

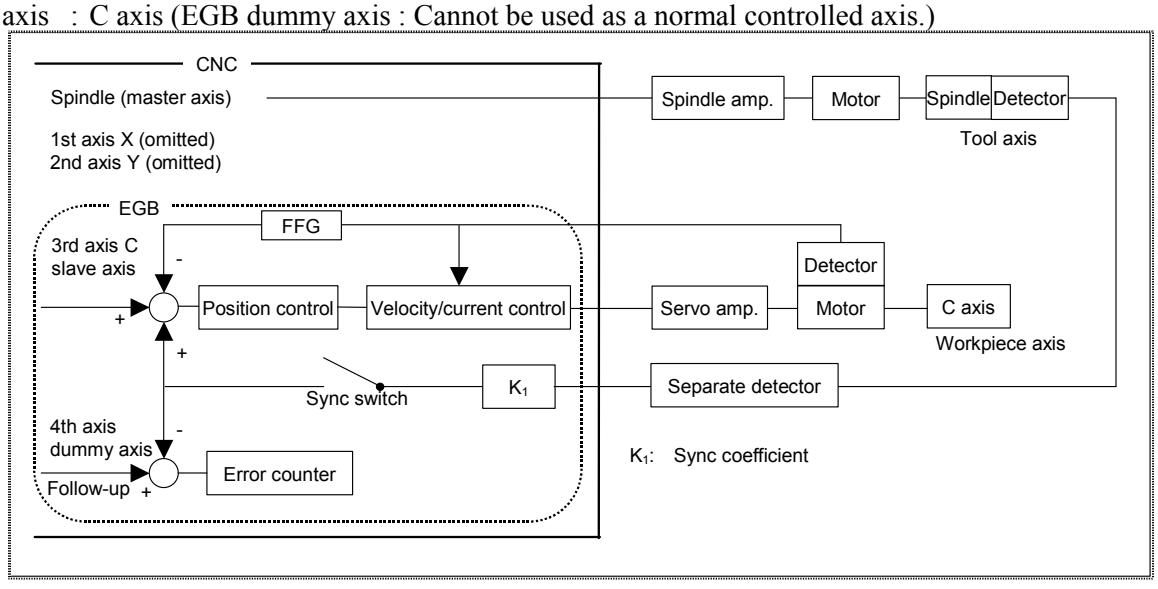
Overview

This function synchronizes the revolutions of the workpiece axis connected to the servo motor with the revolutions of the tool axis (grinding stone/hob) connected to the spindle motor so as to machine (grind/cut) gears as in the hobbing machine function. The synchronization ratio can be specified by a program. Synchronization of the tool axis and the workpiece axis by this function is directly controlled by a digital servo, so the workpiece axis can track changes in the speed of the tool axis with no error, thereby achieving high-precision machining of gears. In the following descriptions, the electric gear box is called the EGB. For details on the conditions that need be met to set the workpiece axis and tool axis, refer to the manual provided by the machine tool builder.

NOTE
Electronic gear box is optional function.

- 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.)



For EGB axis configuration parameter setting examples, see the section on "FSSB setting".

Format

	Parameter EFX(No.7731#0)=0	Parameter EFX(No.7731#0)=1
Start of synchronization	**G81 T__ (L__) (Q__ P__) ;**	**G81.4 T__ (L__) (Q__ P__) ;**
Cancellation of synchronization	**G80 ;**	**G80.4 ;**

T(or R) : Number of teeth (Specifiable range: 1 to 3000)
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)
* 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 synch switch of the EGB function is closed, and the synchronization of the tool and workpiece axes is started. At this time, the EGB mode signal SYNMOD becomes 1. 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 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 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

In synchronism with gradual stop of the tool axis, the workpiece axis is decelerated and stopped. By specifying the command below after the spindle stops, synchronization is canceled, and the EGB synchronization switch is opened. At this time, the EGB mode signal SYNMOD becomes 0.

(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 HBR, bit 0 of parameter No. 7700, to 0, it is possible to cancel synchronization with a reset. If the manual absolute signal is ON, the absolute coordinate is updated.

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.

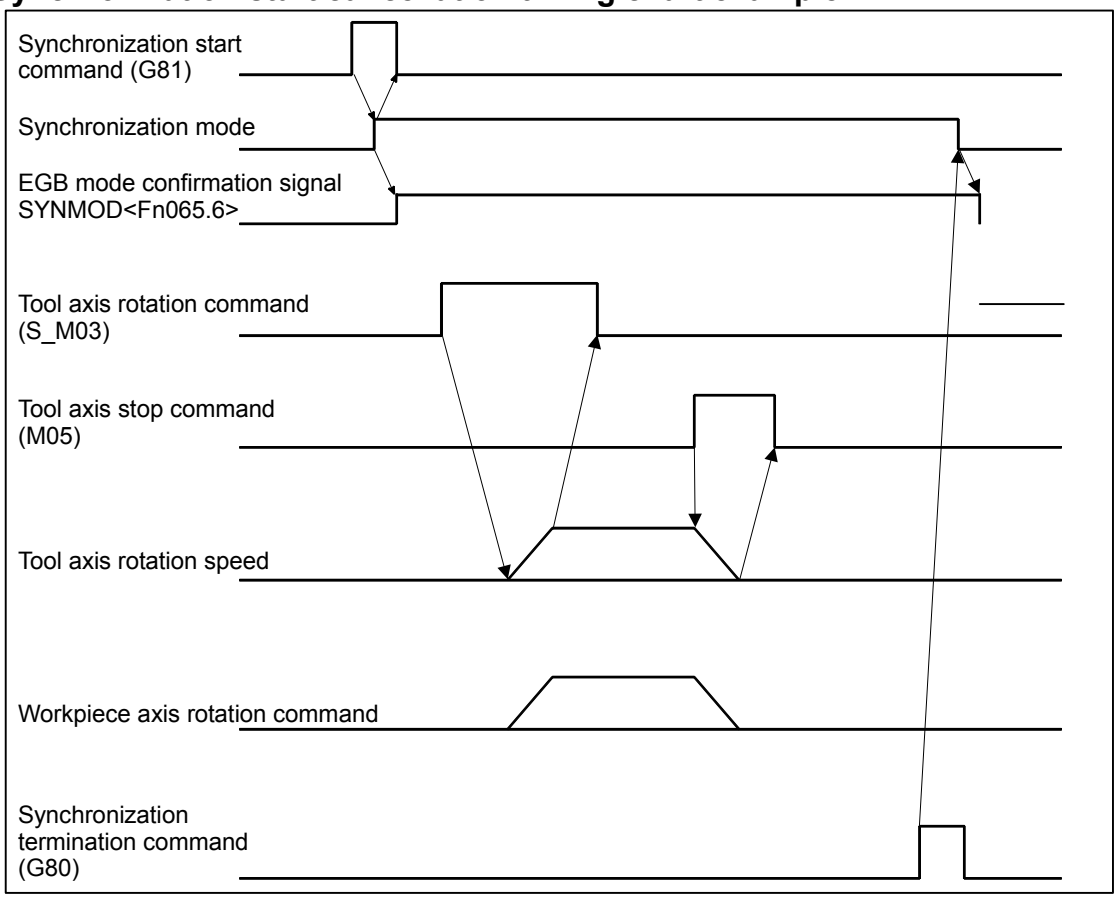
⚠ CAUTION

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, and G53 for a slave axis.
3 It is not possible to use controled 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 ERV, bit 0 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.

- Synchronization start/cancellation timing chart example



- To reduce a synchronous error

When the electronic gear box is used, be sure to make the feed-forward for the slave axis valid and set 100% in the feed-forward factor to reduce a synchronous error.
In addition, confirm whether the feed-forward operates effectively and a synchronous error is reduced by the following procedure.

[Procedure to confirm]

<1> When the slave axis synchronizes only with the command to the master axis (When the helical gear compensation is not performed), the positioning deviation of the slave axis corresponds to a synchronous error. At this time, confirm whether the positioning deviation of the slave axis (diagnosis No.300) is about 0.

<2> Confirm whether the positioning deviation of the slave axis is still about 0 even if the number of rotations of the master axis is changed.

It is necessary to perform the following setting to make the feed-forward valid with the feed-forward factor 100%.

[Setting of parameters]

- Bit 3 (PIEN) of parameter No.2003 for the slave axis is set to 1.
  That is, the PI control is selected as a velocity control method.
- Bit 1 (FEED) of parameter No.2005 for the slave axis is set to 1.
  That is, the feed-forward function is made valid.
- Bit 1 (FFAL) of parameter No.2011 for the slave axis is set to 1.
  That is, the feed-forward control is made valid without depending on the mode.
- Parameter No.2068 for the slave axis is set to 10000.
  That is, the feed-forward factor is set to 100%.

For details of the feed-forward function, refer to the section of "Feed-forward Function" in "CONTOUR ERROR SUPPRESSION FUNCTION" in 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).

- To reduce a mechanical shock at the start of the acceleration/deceleration

Be sure to make the soft start/stop function valid for the spindle axis if the mechanical shock due to acceleration/deceleration is large when the master axis is commanded with the speed mode of the spindle. It is necessary to perform the following setting to make the soft start/stop function valid.

[Setting of parameters]
- Bit 2 (SOSALW) of parameter No.4399 is set to 1.
  That is, even when the emergency stops, the soft start/stop function is made valid.
- Parameter No.4030 is set.
  That is, the soft start/stop setting time is set.
- Parameter No.4508 is set.
  That is, the rate of change in acceleration at soft start/stop is set.

NOTE
About the parameter No.4030 and No.4508, those adjustments matched to the spindle are needed so that mechanical shock is reduced.

[Setting of signals]
- Soft start/stop signal (for the first spindle) <G0071.4>
- Soft start/stop signal (for the second spindle) <G0075.4>

For details of the soft start/stop function, refer to FANUC AC SPINDLE MOTOR α*i*/β*i* series BUILT-IN SPINDLE MOTOR B*i* series PARAMETER MANUAL (B-65280EN).

- 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:

$$\text{Compensation angle} = \frac{Z \times \sin(P)}{\pi \times T \times Q} \times 360 \text{ (for metric input)}$$

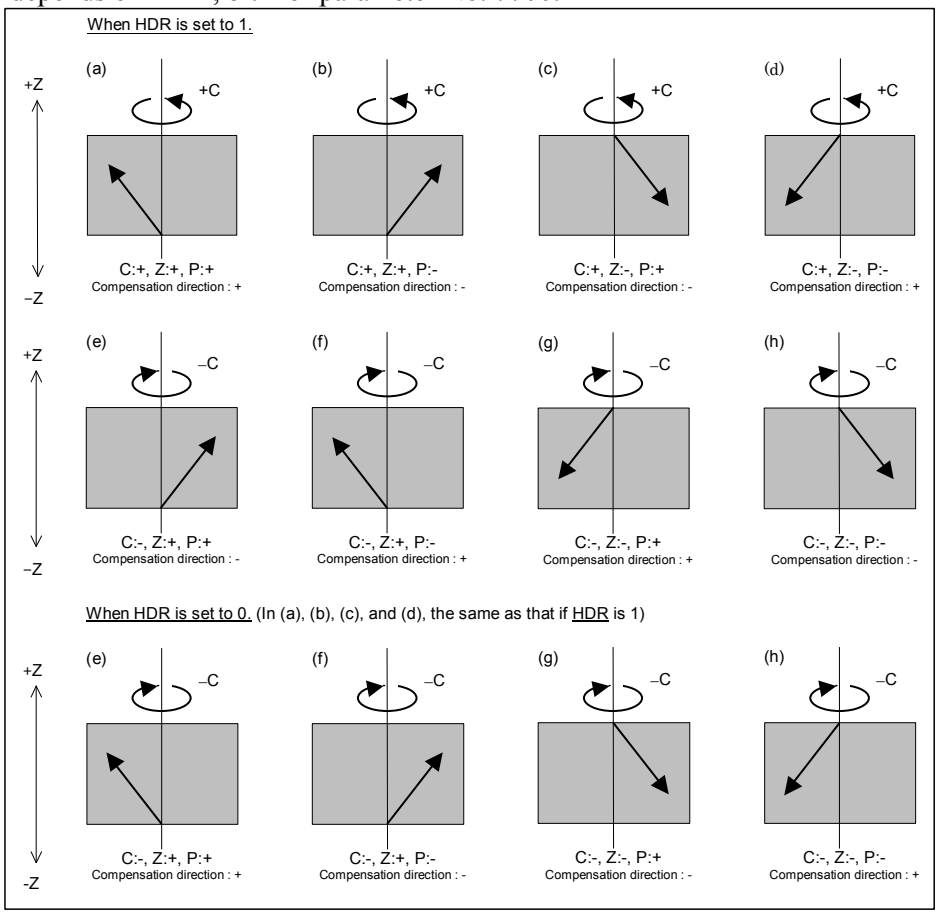
$$\text{Compensation angle} = \frac{Z \times Q \times \sin(P)}{\pi \times T} \times 360 \text{ (for inch input)}$$

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 on the workpiece axis and the absolute coordinates are updated with helical 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:

$-2147483648 \leq K_n \leq 2147483647$
$1 \leq K_d \leq 65535$

When this restriction is not satisfied, the alarm (PS1596) is issued when G81 is specified.

- Retract function

(1) Retract function with an external signal

When the retract signal, RTRCT, becomes "1" (the rise of the signal is captured), retraction is performed with the retract amount set in parameter No. 7741 and the speed set in parameter No. 7740.

No movement is performed along an axis for which 0 is set as the retract amount.

After the end of retraction, the retract completion signal, RTRCTF, is output.

(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.

After the end of retraction, the retract completion signal, RTRCTF, is output.

Conditions under the retract function with an alarm

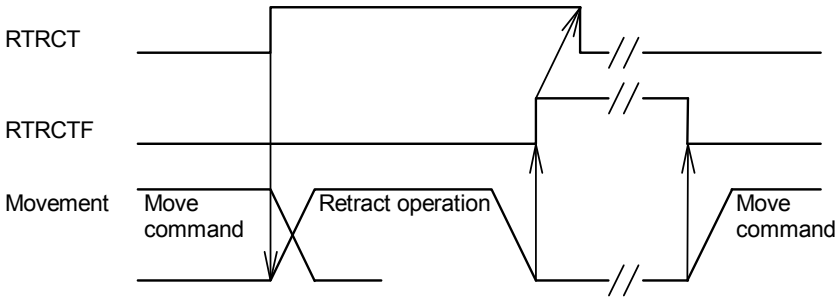
The conditions under which the retract function with an alarm can be changed using the settings of ARE, bit 1 of parameter No. 7703, and bit 2 (ARO) of parameter No. 7703.

The table below lists parameter settings and corresponding conditions.

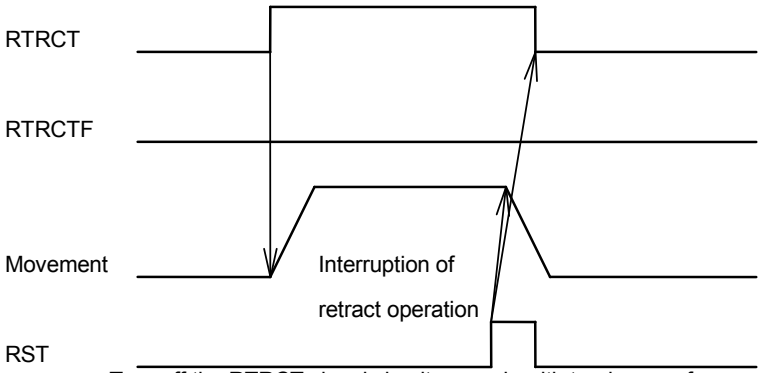
ARE	ARO	Condition
1	0	EGB synchronization is in progress.
1	1	Both EGB synchronization and automatic operation are in progress.
0	0	Either EGB synchronization or automatic operation is in progress.
0	1	

Timing chart

<1> On/off timing of RTRCT and RTRCTF

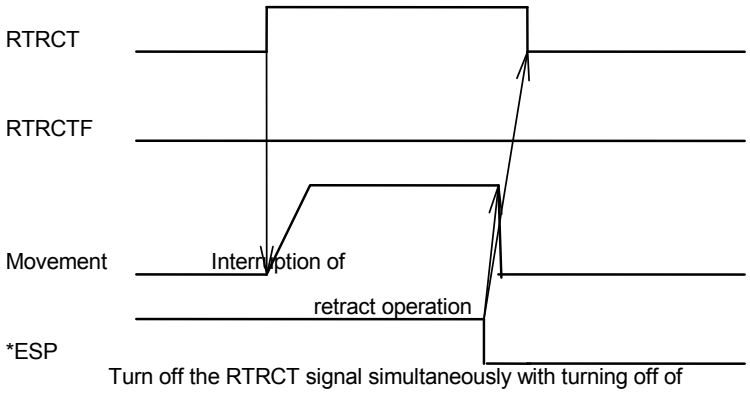


<2> Interruption of retraction with to a reset



Turn off the RTRCT signal simultaneously with turning on of the RST signal.

<3> Interruption of retraction due to an emergency stop



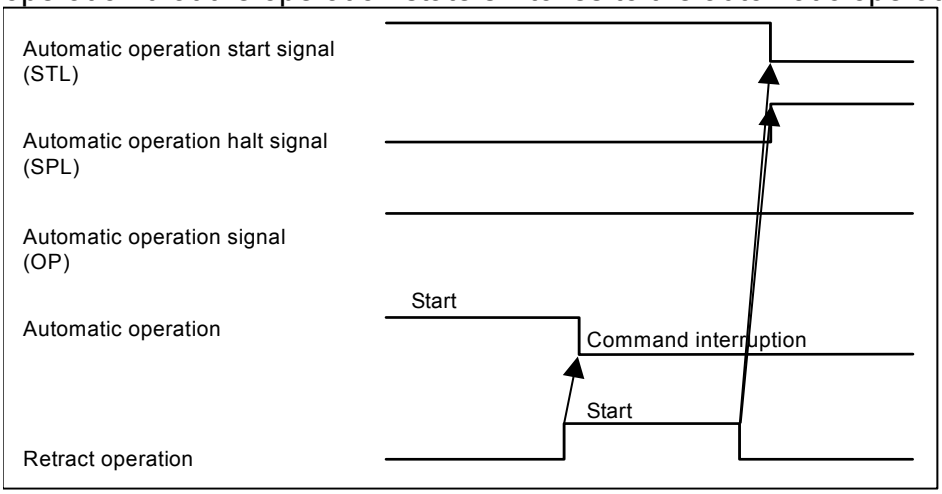
Turn off the RTRCT signal simultaneously with turning off of the *ESP signal.

⚠ 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. The retract operation terminates in the machine lock state, and a retract completion signal is output.
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.

NOTE

8 If, during a retract operation, a reset or an emergency stop is made, the operation is interrupted. At this time, the retract completion signal does not become "1".

9 If an overtravel alarm is issued on a retract axis while a retract operation is in progress on multiple axes, the retract option stops for only the axis where the alarm is issued if bit 4 (RTS) of parameter No. 7731 is 0. If bit 4 of parameter No. 7731 is 1, the retract operation is interrupted for all the axes. If a servo alarm is issued, the retract operation is interrupted for all the axes regardless of the setting of bit 4 (RTS) of parameter No. 7731. At this time, the retract completion signal does not become "1".

10 To enable the retract function with an alarm, bit 3 (ART) of parameter No.7702, must be set.

11 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 geneated on the retract axis.

12 If a new alarm is issued during retraction with the retract function with an alarm, a retract operation is not performed.

13 After the end of retraction with a servo alarm, servo position control stops in 400 ms.

14 The retract completion signal, RTRCTF, becomes "0" when a move command is issued for any axis after retract operations on all axes are completed.

Example

```
O1000 ;
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 work axis
                         (Rotation about the work 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 (cutting)
N0060 G01 Z_ F_ ;      Movement along the Z-axis (machining)
 ---------------------- ;  If required, C, X, Z, 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 work axes
N0130 M30 ;
```

Signal

Retract signal RTRCT <Gn066.4>

[Classification] Input signal

[Function] Performs retraction for the axis specified with a parameter.

[Operation] When this signal becomes "1", the CNC operates as follows:

The CNC can capture the rise of the signal and perform retraction for the axis for which the retract amount has been specified with parameter No. 7741. The retract amount and the retract speed will be the values previously set in parameters Nos. 7741 and 7740. After the end of retraction, the retract completion signal, RTRCTF, is output. The retract signal is effective both during automatic operation mode (MEM, MDI, etc.) and manual operation mode (HNDL, JOG, etc.). If, during automatic operation, the retract signal is set to "1", a retract operation is performed and, at the same time, automatic operation is halted.

Retract completion signal RTRCTF <Fn065.4>

[Classification] Output signal

[Function] Posts notification of the completion of retraction.

[Operation] This signal is set to 1 in the following case:

- Upon the completion of retraction (at the end of movement)

This signal is set to 0 in the following case:

- When a move command is issued for any retract axis after the end of a retract operation.

NOTE
The retract signal is not accepted while the retract completion signal is set to 1.

EGB mode signal SYNMOD <Fn065.6>

[Classification] Output signal

[Function] Posts notification that synchronization using the EGB is in progress.

[Operation] This signal is set to 1 in the following case:

- While synchronization using the EGB is in progress

This signal is set to 0 in the following case:

- Once synchronization using the EGB has terminated

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn066				RTRCT				
Fn065		SYNMOD		RTRCTF				

Parameter

The table below gives parameters related to EGB.

Parameter number	Description
1006 # 0 1006 # 1	An EGB slave axis and an EGB dummy axis require that the setting of a rotary axis (type A) (bit 0 (ROT) of parameter No. 1006 be 1 and bit 1 (ROS) of parameter No. 1006 be 0.
1023	Set from the FSSB setting screen. For FSSB manual setting, be sure to set the EGB axis as described below: The slave axis must be set with an odd number, and the dummy axis with an even number. They must be consecutive. Example: If the servo axis number of the slave axis is "1", the servo axis number of the dummy axis must be set to "2". If the servo axis number of the slave axis is "3", the servo axis number of the dummy axis must be set to "4".
2011 #0	Specify an axis to be synchronized. Specify 1 for both an EGB slave axis and EGB dummy axis.
3115 # 0	The current position is not indicated for an axis for which this parameter is set to 1.Since the current position for an EGB dummy axis has no meaning, set this parameter to 1 to delete the current position indication for the axis from the screen.
7700 # 0	The synchronous mode is canceled (0)/not canceled (1) by a reset.
7700 # 2	Compensation direction for helical gear compensation
7701# 3	At the start of synchronization (G81), synchronization is started (0)/not started (1) if the number of hob threads L is specified as 0.
7702 # 0	The specifiable number of teeth, T, at the start of synchroniztion (G81) is not reduced to a 1/10 of a specified value (0)/reduced (1).
7702 # 3	The retract function with an alarm is disabled (0)/enabled (1).
7703 # 0	During synchronization (G81), feed per revolution is performed for feedback pulses (0)/pulses converted to the speed for the workpiece axis(1).

Parameter number	Description
7703 # 1,2	Specify when to perform a retract operation with the retract function with an alarm; during synchronization; during synchronization and automatic operation; or during synchronization or automatic operation.
7709	Number of the axial feed axis in helical compensation
7731#0	The EGB command is G80 and G81(0)/G80.4 and G81.4(1).
7731 # 3	When the automatic phase synchronization function for the electronic gear box is disabled, the G81 command cannot be issued again (an alarm is issued) (0)/can be issued again (1)during EGB synchronization.
7731 # 4	When an OT alarm or axis type malfunction protection alarm is issued during EGB retract operation, only the axis for which the alarm is issued is stopped (0)/all axes are stopped (1).
7731 # 5	In EGB synchronization start command G81.4, the number of teeth is specified in T (0)/specified in R (1).
7740	Retraction speed
7741	Retraction amount
7772	Number of position detector pulses per rotation about tool axis
7773	Number of position detector pulses per rotation about workpiece axis

For FSSB settings, see the section on "FSSB settings".
If FSSB setting mode is automatic setting mode, setting is made automatically by inputting data to the FSSB setting screen. For the slave/dummy axes of EGB, set the value in the 'M/S' item in the FSSB axis setting screen same way of the tandem setting.
Note the following points when specifying parameters for the electronic gear box.

1 Specify an axis that is not used or the same name as that for a slave axis for the name of a dummy axis. Do not use a name which is usually not allowed to be used as an axis address, such as D.
2 Specify the same values for an EGB slave axis and an EGB dummy axis in the following parameters.
  1013#0 to 3 Increment system
  1004#7 Ten times minimum input increment
  1006#0,1 Rotary axis setting
  1006#3 Diameter/radius specification
  1420 Rapid traverse rate
  1421 Rapid-traverse override F0 speed
  1820 Command multiplication
  2000 and over Parameters related to digital servo
3 Specify the amount of travel per rotation about a rotation axis for a slave axis and dummy axis in a parameter No. 1260.
4 Make the specification for a dummy axis in the following way.
  1815#1 Whether to use separate detectors. Although an EGB dummy axis uses the interface of a separate detector, set these parameters to 0.
5 Be sure to make the feed-forward for the slave axis valid to reduce a synchronous error.
  For details, refer to “To reduce a synchronous error” in the explanation of this section.
6 Be sure to make the soft start/stop function valid for the spindle axis to reduce a mechanical shock at the start of the acceleration/deceleration.
  For details, refer to “To reduce a mechanical shock at the start of the acceleration/deceleration” in the explanation of this section.

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 Number of controlled axes

Set the servo axis for each control axis.
Usually set to same number as the control axis number.
The control axis number is the order number that is used for setting the axis-type parameters or axis-type machine signals

* For electronic gear box (EGB) controlled axes, two axes need to be specified as one pair. So, make a setting as described below.
EGB axis:
For a slave axis, set an odd (1, 3, 5, 7, ...) 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.

	#7	#6	#5	#4	#3	#2	#1	#0
2011								SYN

[Input type] Parameter input
[Data type] Bit axis

#0 SYN 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 the slave and dummy axes of the EGB.

NOTE
For the setting of this parameter to take effect, the power must be turned off and back on again.

	#7	#6	#5	#4	#3	#2	#1	#0
3115								NDPx

[Input type] Parameter input
[Data type] Bit axis

#0 NDPx The current position display is:
0: performed.
1: not performed.

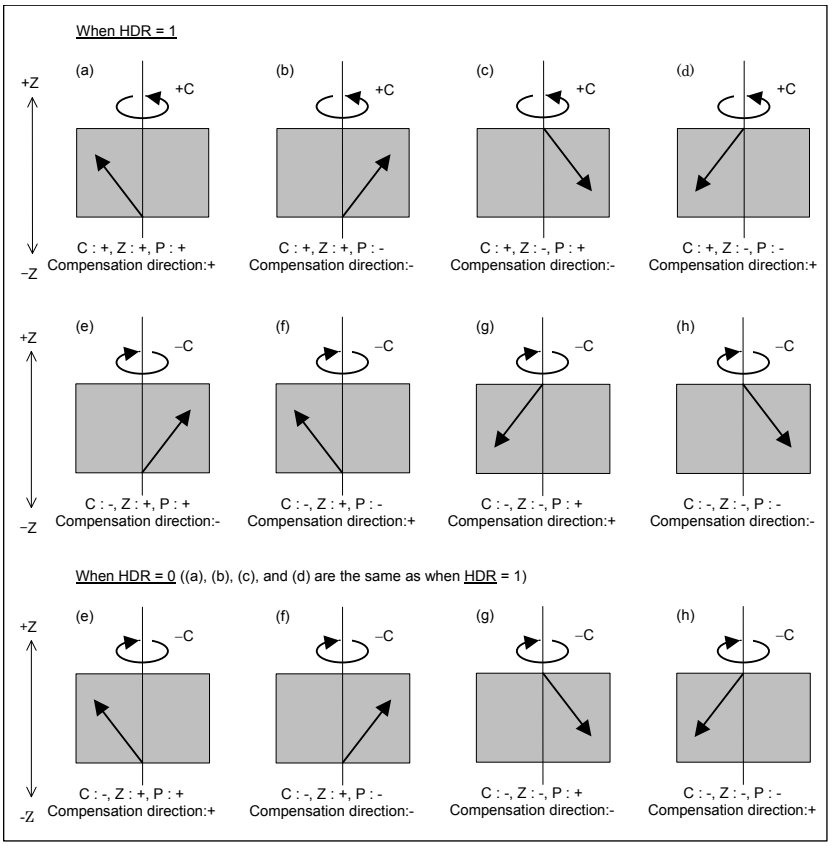
NOTE
If using the electronic gear box function (EGB), set 1 for the dummy axis of EGB to suppress position display.

	#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 synchronous mode (G81).
1: Does not cancel the synchronous mode. The mode is canceled only by the G80 command.

#2 HDR Direction for compensation for a helical gear (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								
					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 (parameter 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.

	#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 as follows:
0: During EGB synchronization or automatic operation (automatic operation signal OP = 1).
1: According to the setting of bit 2 (ARO) of parameter No. 7703

#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).

NOTE
This parameter is valid when bit 1 (ARE) of parameter No. 7703 is set to 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
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 is enabled).

7709	Number of the axial feed axis for helical compensation

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to Number of controlled axes
This parameter sets the number of the axial feed axis for a helical gear.

NOTE
When this parameter is set to 0 or a value outside the valid setting range, the Z-axis is used as the axial feed axis.

	#7	#6	#5	#4	#3	#2	#1	#0
7731								
				RTS	ECN		EHF	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.

#1 EHF Feed-forward control for the axial feed axis for helical compensation is:
0: Enabled only during cutting.
1: Always enabled in the G81 synchronous mode.
Usually, set 0.
Feed-forward control is usually enabled in the cutting feed mode. When this parameter is set to 1, feed-forward control is always enabled for the axial feed axis for helical compensation during synchronization by the command (G81) for a hobbing machine.
When bit 3 (FFR) of parameter No. 1800 is set to 1, feed-forward control is always enabled regardless of the setting of this parameter.

#3 ECN When the automatic phase synchronization function for the electronic gear box is disabled, during EGB synchronization, the G81 command:
0: Cannot be issued again. (The alarm (PS1595) is issued.)
1: Can be issued again.

#4 RTS When an OT alarm or axis type malfunction protection alarm is issued during EGB retract operation:
0: Only the axis for which the alarm is issued is stopped.
1: All axes are stopped.

7740	Feedrate during retraction

[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 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.

7741	Retracted distance

[Input type] Parameter input
[Data type] Real axis
[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)
This parameter sets the retracted distance for each axis.

7772	Number of position detector pulses per rotation about the tool axis

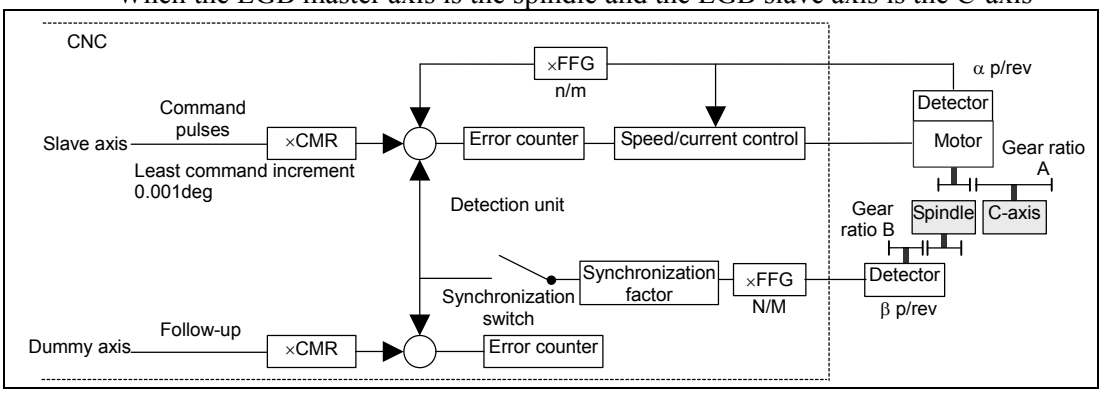
[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Detection unit
[Valid data range] 1 to 999999999
This parameter sets the number of pulses per rotation about the tool axis (on the spindle side), for the position detector.
For an A/B phase detector, set this parameter with four pulses equaling one A/B phase cycle.

NOTE
Set the feedback pulse of position detector pulses per rotation about the tool axis including gear ratio of the spindle to position coder.

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 example.

	#7	#6	#5	#4	#3	#2	#1	#0
2273		EGF						

[Input type] Parameter input
[Data type] Bit axis

#6 EGF FFG is:
0: Not considered in the synchronization coefficient.
1: Considered.

The synchronization coefficient is subject to the following restriction:
Synchronization coefficient $= \frac{L}{T} \times \frac{\beta}{\alpha}$
where $\frac{L}{T} \times \frac{\beta}{\alpha} \leq \frac{2\text{word}}{1\text{word}}$: Condition <1>
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)

If this condition, <1>, cannot be satisfied, set this parameter bit to 1. With this setting, FFG is considered in the synchronization coefficient, and by selecting FFG appropriately, it is possible to set α and β in such a way that condition <1> can be satisfied with the synchronization coefficient kept intact.

Synchronization coefficient $= \frac{L}{T} \times \frac{\beta}{\alpha} \times \frac{N}{M}$

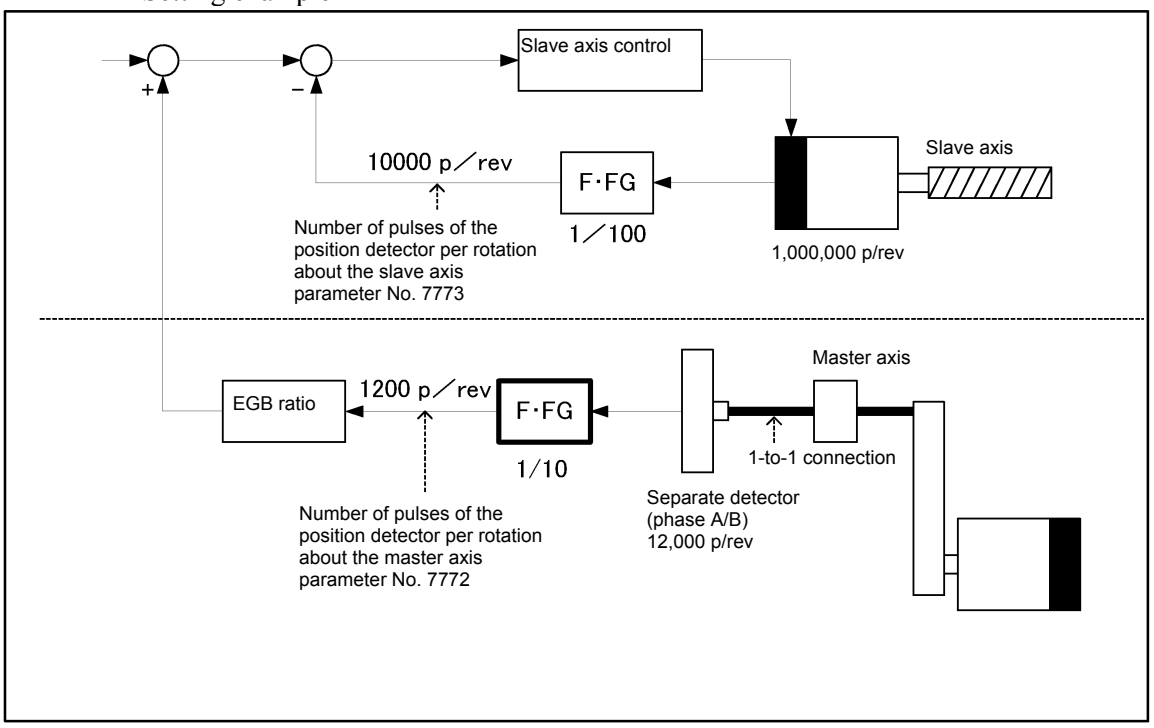
where $\frac{L}{T} \times \frac{\beta}{\alpha} \leq \frac{2\text{word}}{1\text{word}}$: Condition <1>

N: Numerator of FFG
M: Denominator of FFG

The new value of α is the old one multiplied by FFG.

$$\alpha[\text{ New }] = \alpha[\text{Old}] \times \frac{N}{M}$$

Setting example



Master axis conditions:
The separate detector must be 12000 p/rev.
The master axis and the separate detector must have a 1-to-1 connection.
Slave axis conditions:
The motor pulse coder must be 1 million p/rev.
FFG must be 1/100.

Determine FFG so that condition <1> is satisfied.

$\frac{L}{T} \times \frac{\beta}{\alpha} \leq \frac{2\text{word}}{1\text{word}}$: Condition <1>

In this example, FFG is set to 1/10. Set bit 6 (EGF) of parameter No. 2273, which is a function bit to consider FFG in EGB, to 1, and set the number of pulses of the position detector per rotation about the master and slave axes.

As the number of pulses of the position detector per rotation about the master axis, set 12000 x FFG (1/10) = 1200.
As the number of pulses of the position detector per rotation about the slave axis, set 10000.

NOTE
Numbers 2000 and over are for the parameters related to digital servo. For details, refer to FANUC AC SERVO MOTOR αi series PARAMETER MANUAL (B-65270EN).

2372	Serial EGB exponent specification (γ)

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 15

By setting a value in this parameter, it is possible to internally multiply the value of parameter No. 7772 or 7782 by 2^{γ}.

With a high resolution serial detector, the number of pulses per rotation is large, causing the denominator of the synchronization coefficient (K_1) to become large, which may fall outside the valid range. By providing 2's exponent component for the number-of-pulses setting per rotation about the workpiece axis, it is possible to keep the denominator of the EGB coefficient low.

• Valid range of a synchronization coefficient
A synchronization coefficient is internally represented using a fraction number (K_n/K_d) to eliminate an error. The formula below is used for calculation.

$$\text{Synchronization coefficient} = \frac{K_n}{K_d} = \frac{L}{T} \times \frac{\beta}{\alpha}$$

L: Number of hob threads
T: Number of teeth
α: Number of pulses of the position detector per rotation about the EGB master axis (parameter No. 7772)
β: Number of pulses of the position detector per rotation about the EGB slave axis (parameter No. 7773)

K_n/K_d is a value resulting from reducing the right side of the above formula, but the result of reduction is subject to the following restrictions:
$-2147483648 \leqq K_n \leqq 2147483647$
$1 \leqq K_d \leqq 65535$
If this condition is not satisfied, an alarm is generated when G81 is issued.

If the number of pulses per rotation about the master axis is large, this condition may not be satisfied. In such a case, use this parameter.

If using a serial detector, set the number of post-FFG pulses. If the exponent specification is used, $\alpha \times 2^{\gamma}$ means the "number of pulses of the position detector per rotation about the master axis".

If a serial type detector is used as the master axis detector, the relationship between master axis feedback (in detector pulse units) and the move command to the slave axis (Detection unit in the NC) is as follows:
(Slave axis move command)

$$= \frac{L}{T} \times \frac{\beta}{\alpha \times 2^{\gamma}} \times \frac{N}{M} \times \text{(master axis feedback)}$$

Setting example)

Number of pulses of the position detector per rotation about the master axis = 1,000,000 [pulse/rev]
Master axis FFG=1/1
Number of pulses of the position detector per rotation about the slave axis = 360,000 [pulse/rev]
Slave axis Detection unit1/1000 [deg]
Then, from 1,000,000 = 15,625 x 2^6, the settings are
α = 15,625, β = 360,000, γ = 6, FFG = N/M = 1/1

NOTE
Numbers 2000 and over are for the parameters related to digital servo. For details, refer to FANUC AC SERVO MOTOR αi series PARAMETER MANUAL (B-65270EN).

Alarm and message

Number	Message	Description
PS1593	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. (ROT, bit 0 of parameter No. 1006) (3) Number of pulses per rotation (parameter No. 7772 or 7773) is not set.
PS1594	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) In the G81 block, only one of P and Q is specified.
PS1595	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, G33, G53, etc. (2) Inch/metric conversion command using G20, G21, etc.
PS1596	EGB OVERFLOW	An overflow occurred in the calculation of the synchronization coefficient.

Reference item

Manual name	Item name
Operator's Manual (M series) (B-64304EN-2)	Electronic gear box

2 PREPARATIONS FOR OPERATION

Chapter 2, “PREPARATIONS FOR OPERATION”, consists of the following sections:

2.1 EMERGENCY STOP 177
2.2 CNC READY SIGNALS 179
2.3 OVERTRAVEL CHECK 180
2.4 ALARM SIGNALS 206
2.5 START LOCK/INTERLOCK 210
2.6 MODE SELECTION 216
2.7 STATUS OUTPUT SIGNAL 222
2.8 VRDY OFF ALARM IGNORE SIGNAL 223
2.9 UNEXPECTED DISTURBANCE TORQUE DETECTION FUNCTION 225
2.10 MACHINING CONDITION SELECTION FUNCTION 235
2.11 MACHINING QUALITY LEVEL ADJUSTMENT (M Series) 241
2.12 MALFUNCTION PREVENT FUNCTIONS 243
2.13 OPERATOR ERROR PREVENT FUNCTIONS 246

2.1 EMERGENCY STOP

Overview

If you press Emergency Stop button on the machine operator's panel, the machine movement stops in a moment.



Fig. 2.1 (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.

Signal

Emergency stop (input) *ESP<X008.4, Gn008.4>

[Classification] Input signal

[Function] Activating an emergency stop signal stops the machine instantly.

[Operation] When the emergency stop signal *ESP turns to "0", the emergency stop is applied to the machine and the CNC is reset. This signal is controlled by the B contacts of a pushbutton switch. The emergency stop signal turns the servo ready signal (SA) to "0".
Overtravel detection by this CNC is handled by the stored stroke check function, and a limit switch for normal overtravel detection is not needed.
To prevent the machine from moving beyond the software limit through servo feedback error, always install a stroke end limit switch (shown in Fig. 2.1 (b) as follows).

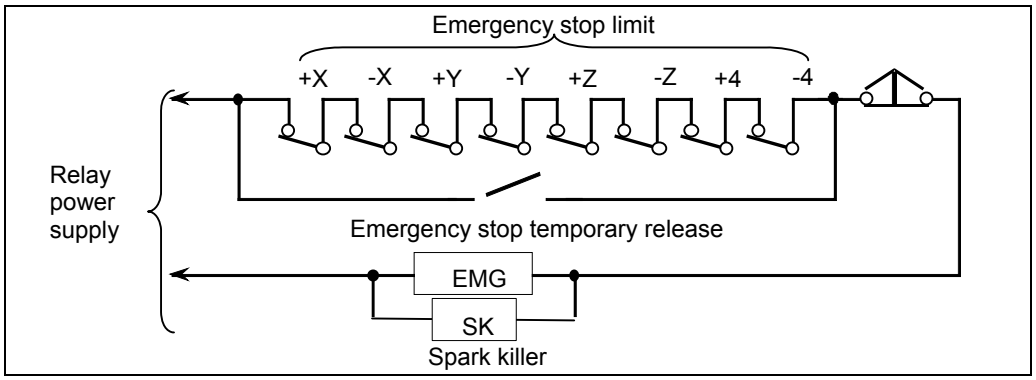
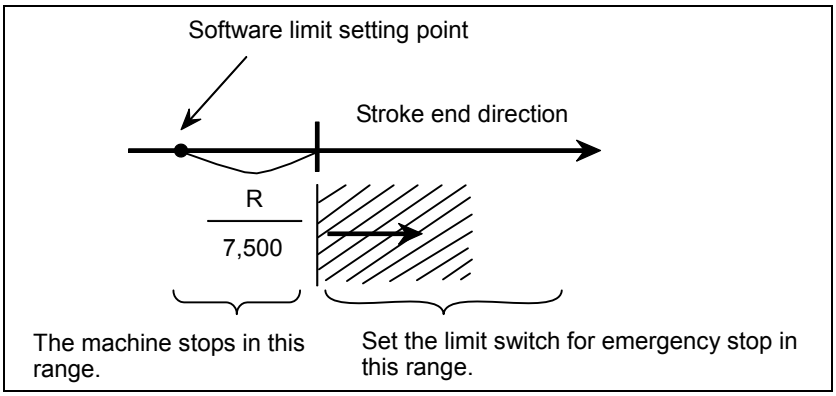


Fig. 2.1 (b) Connection of emergency stop limit switch

The distance from the position where the dynamic brake is applied to that where the tool stops moving is given in the "AC Servo Motor αi Descriptions (B-65262EN)."

> ⚠ **WARNING**
> Software limit setting point and operating point of limit switch for emergency stop
> The stop point by the software limit goes beyond the setting point by as much as the following distance.
> The actual stopping point may exceed the position set by a parameter (Nos.1320 and 1321) by as much as R/7500 (mm).
> R/7,500 (mm)
> R: Rapid traverse rate (mm/min)
> Set the limit switch for emergency stop including the allowance for the above value.



Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X008				*ESP				
	#7	#6	#5	#4	#3	#2	#1	#0
Gn008				*ESP				

Reference item

Manual name	Item name
FANUC AC SERVO MOTOR αi series DESCRIPTIONS (B-65262EN)	Servo motor selection

2.2 CNC READY SIGNALS

Overview

When the CNC is turned on and becomes ready for operation, the CNC ready signal is set to 1.

Signal

CNC Ready Signal MA<Fn001.7>

[Classification] Output signal

[Function] The CNC ready signal indicates that the CNC is ready.

[Output cond.] When the CNC is turned on and becomes ready for operation, the signal is set to 1. Normally, it takes several seconds to establish this state after the power is turned on. If a system alarm is issued, the signal is set to 0. The signal remains set to 1, however, when an emergency stop or a similar operation is performed.

Servo Ready Signal SA<Fn000.6>

[Classification] Output signal

[Function] Signal SA turns to "1" when the servo system is ready to operate. For an axis that is to be braked, release the brake when this signal is "1" and apply the brake when this signal is "0".

Time chart of this signal is as follows:

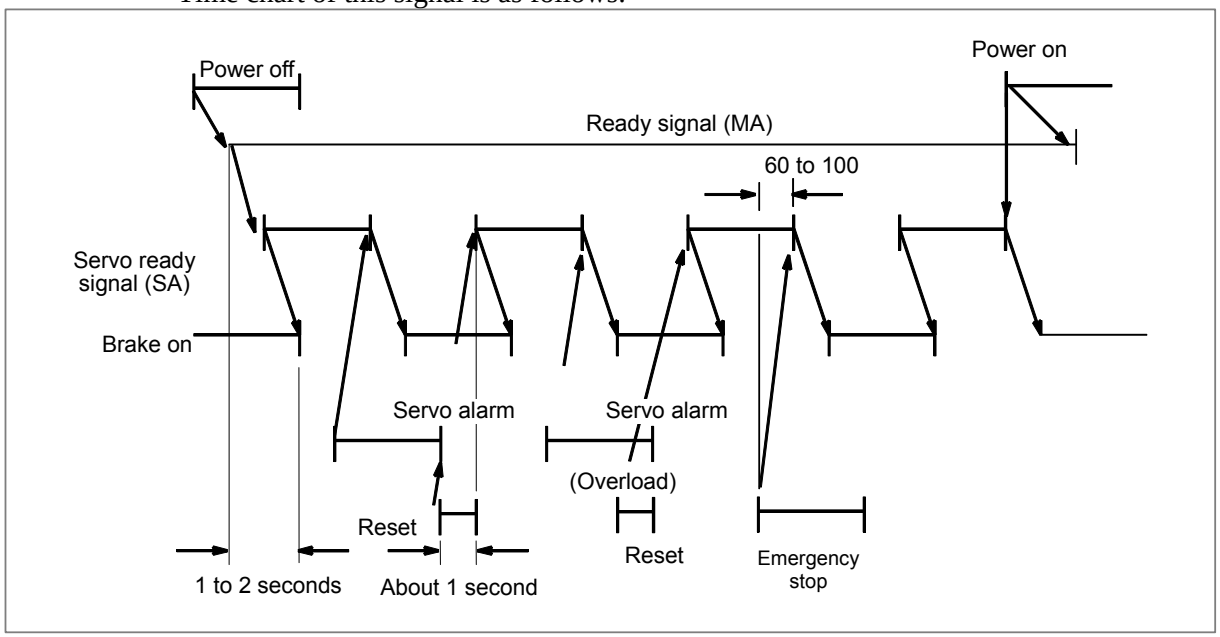


Fig. 2.2 (a) Time chart for servo ready signal

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn000		SA						
Fn001	MA							

2.3 OVERTRAVEL CHECK

2.3.1 Overtravel Signals

Overview

When the tool tries to move beyond the stroke end set by the machine tool limit switch, the tool decelerates and stops as a result of tripping the limit switch, and an OVER TRAVEL is displayed.

Signal

Overtravel signals *+L1 to *+L5<Gn114.0 to ~Gn114.4>,*-L1 to *-L5<Gn116.0 to Gn116.4>

[Classification] Input signal

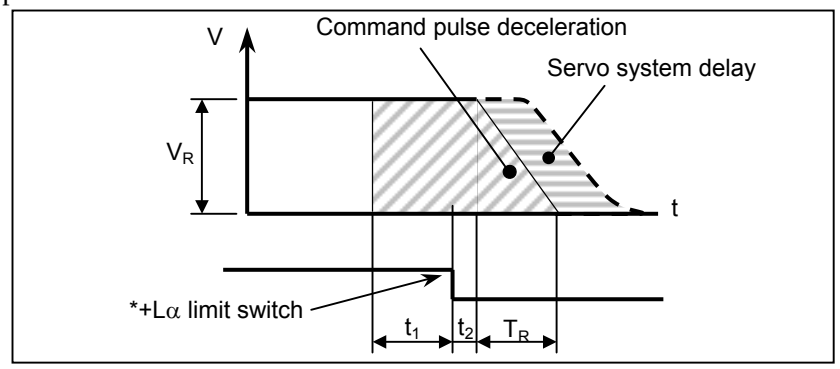
[Function] Indicates that the control axis has reached its stroke limit. There are individual signals for each direction in every control axis.
The +/- in the signal name indicates the direction and the number corresponds to the control axis.
+Lx
x : 1 : No. 1 axis is at stroke limit.
2 : No. 2 axis is at stroke limit.
:
:
5 : No. 5 axis is at stroke limit.
+ : Limit reached in + direction.
- : Limit reached in - direction.

[Operation] When it is "0" , the control unit operates as given below.

- In automatic operation, if even one axis overtravel signal turns to "0", all axes are decelerated to stop, an alarm is given and operation is halted.
- In manual operation, only the axis whose overtravel signal has turned to "0" is decelerated to a stop, and the axis can be moved in the opposite direction.
- Once the axis overtravel signal has turned to "0", the axis direction is registered. Even if the signal returns to "1", it is not possible to move that axis in that direction until the alarm is cleared.

The following shows the deceleration distance at overtravel.

(i) Rapid traverse



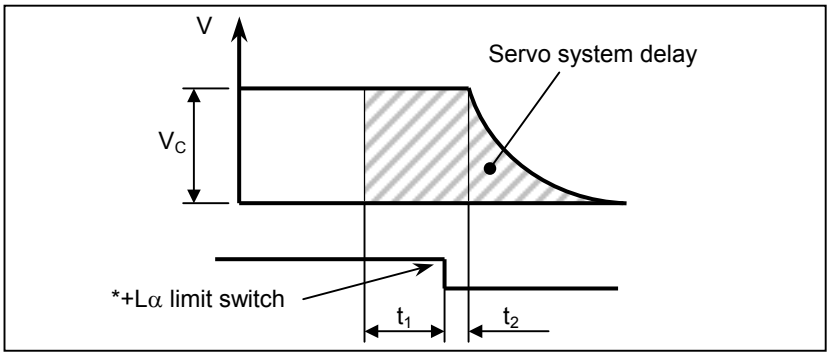
$L_1=V_R(t_1+t_2+T_R/2+T_S)\times 1/6000$ [mm or inch].....<1>

L_1 : Deceleration distance
V_R : Rapid traverse speed [mm/min or inch/min]
t_1 : Limit switch signal delay time (from limit switch operation to *+La signal turn off [msec])

t_2 : Receiver delay time 30msec
T_R : Rapid traverse acceleration/deceleration time constant [msec]
T_S : Servo system time constant [msec]

NOTE
Servo system time constant T_S is 33 msec when the servo unit is adjusted to the standard setting.

(ii) Cutting feed



V_C : Maximum feedrate [mm/min or inch/min]
L_2 : Deceleration distance
t_1, t_2, Ts: Same as (i).
$L_2=V_C(t_1+t_2+T_R/2+T_S)\times 1/6000$ [mm or inch].....<2>

Releasing overtravel

First, move the tool into a safe zone under manual operation. Then press the reset button to reset the alarm.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn114				*+L5	*+L4	*+L3	*+L2	*+L1
Gn116				*-L5	*-L4	*-L3	*-L2	*-L1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3004			OTH					

[Input type] Parameter input
[Data type] Bit path

#5 OTH The overtravel limit signal is:
0: Checked
1: Not checked

⚠ WARNING
For safety, usually set 0 to check the overtravel limit signal.

Alarm and message

Number	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.

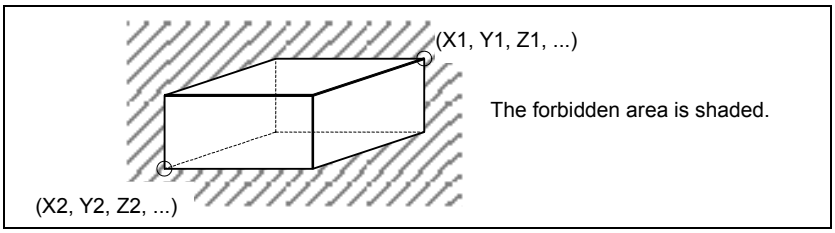
Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Overtravel

2.3.2 Stored Stroke Check 1

Overview

A machine movable range is set with coordinates in the machine coordinate system in parameters. If the machine attempts to move beyond the range, it is decelerated and stopped and an alarm is displayed.
This function is enabled after manual reference position return is performed after power-on.
It can be used instead of an overtravel limit switch (hardware component).
When both functions are used, both are valid.
Unlike an overtravel limit switch, this function checks whether the position at which the machine is stopped after decelerated from the current position is beyond the range.
The stroke check 1 release signal common to all axes can be set to 1 so that the control unit does not make stroke check 1.



Explanation

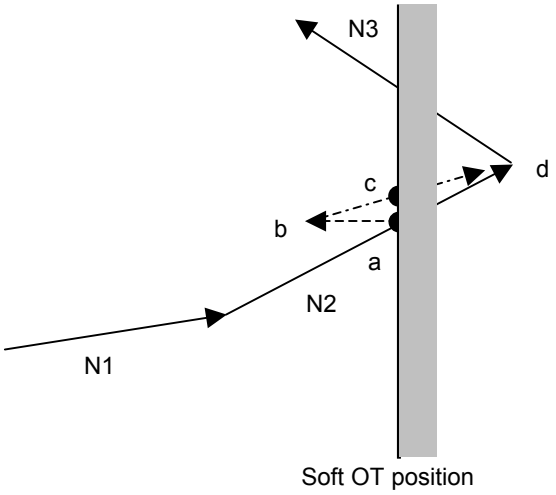
Parameters (Nos. 1320, 1321 or Nos. 1326, 1327) set boundary. Outside the area of the set limits is a forbidden area. The machine tool builder usually sets this area as the maximum stroke.
When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction 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.

Automatically released alarm

If bit 4 (OF1) of parameter No. 1301 is set to 1,when the tool moves along an axis and goes out of the overtravel area, the alarm can be released without performing a reset operation.

NOTE

1 When a soft OT1 alarm is issued during automatic operation, the soft OT alarm may be issued each time operation restarts after the tool is moved along the aixs to a position within the stroke limits through manual intervention, which can prevent the operation from being continued.
Example:
Suppose that a path N1→N2→N3 is programmed as shown in the figure below and that an absolute command in the N2 block causes a soft OT alarm at point a. When automatic operation is restarted after the soft OT alarm is released automatically by retracting the tool to point b by manual intervention in the manual absolute mode, the tool moves toward point d, which is the end point of the N2 block, so the soft OT alarm is issued again at point c. It is impossible to proceed to the N3 block.



2 The alarm can also be released automatically when the tool has entered the movable range by moving the tool along an axis by applying external force to the tool in the emergency stop state or servo-off state.

Signal

Stored stroke check 1 select signal EXLM<Gn007.6>

[Classification] Input signal

[Function] Selects stroke check 1-I (parameter Nos. 1320 and 1321) or stroke check 1-II (parameter Nos. 1326 and 1327).

[Operation] When this signal is set to 1, checks stroke check 1 on the basis of parameter Nos. 1326 and 1327,instead of parameter Nos. 1320 and 1321. The parameter values are used for all axes.
If the bit 2 (LMS) of parameter No.1300 is set to 1, this signal is enabled.

NOTE
This signal is invalid when bit 0 (DLM) of parameter No. 1301 is set to 1.

Stroke check release signal RLSOT<Gn007.7>

[Classification] Input signal

[Function] Selects whether the stored stroke check 1 limits are checked or not.

[Operation] When this signal is set to 1, does not check the stored stroke check 1 limits.

Axis direction dependent stored stroke check 1 switch signal +EXL1 to +EXL5<Gn104.0 to Gn104.4>, -EXL1 to -EXL5<Gn105.0 to Gn105.4>

[Classification] Input signal

[Function] Switches between stroke limit 1-I (parameter No. 1320 and No. 1321) and stroke limit 1-II (parameter No. 1326 and No. 1327) for each axis direction.
When this signal goes 1, the CNC operates as described below.

1 +EXL1, +EXL2, . . .
Stroke check 1 (+ side) is checked using parameter No. 1326 instead of No. 1320.

2 -EXL1, -EXL2, . . .
Stroke limit 1 (- side) is checked using parameter No. 1327 instead of No. 1321.

If the bit 0 (DLM) of parameter No.1301 is set to 1, this signal is enabled. At this time, stored stroke limit switch signal EXLM<Gn007.6> is disabled.

Overtravel alarm signal +OT1 to +OT5<Fn124.0~Fn124.4>, -OT1 to -OT5<Fn126.0~Fn126.4>

[Classification] Output signal

[Function] Indicates whether the tool was about to enter the parameter-specified forbidden area (stored stroke limits).
This signal is valid when bit 1 (NAL) of parameter No. 1300 is set to 1 or when bit 6 (OTS) of parameter No. 1301 is set to 1. For differences in output condition between these parameter settings, see [Output cond.] below.
This signal is provided for each direction of each controlled axis. The sign + or - in the signal name indicates the direction, and the number at the end of the signal name indicates the number of a controlled axis.

* xOTy

y: 1 ….. Decelerates the first axis.
2 ….. Decelerates the second axis.
3 ….. Decelerates the third axis.
4 ….. Decelerates the fourth axis.
5 ….. Decelerates the fifth axis.

x: +: Positive direction
-: Negative direction

[Output cond.] If bit 1 (NAL) of parameter No. 1300 is set to 1, this signal is set to 1 when a move command in automatic operation or manual operation causes the tool to be about to enter the forbidden area (stored stroke limit 1) specified by parameter Nos. 1320 and 1321.
If bit 6 (OTS) of parameter No. 1301 is set to 1, this signal is set to 1 when one of the following overtravel alarms is issued:

- Alarms OT0500, OT0501 : Stored stroke check 1
- Alarms OT0502, OT0503 : Stored stroke check 2
- Alarms OT0504, OT0505 : Stored stroke check 3
- Alarms OT0506, OT0507 : Hard OT
- Alarms OT0510, OT0511 : Stored stroke check before movement

Once the signal is set to 1, the axis and direction of the signal are stored. It is impossible to further move the tool along that axis in that direction. The tool can be moved along the axis only in the direction opposite to the stored direction. Moving the axis in the opposite direction resets the signal to 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn007	RLSOT	EXLM						
Gn104				+EXL5	+EXL4	+EXL3	+EXL2	+EXL1
Gn105				-EXL5	-EXL4	-EXL3	-EXL2	-EXL1
Fn124				+OT5	+OT4	+OT3	+OT2	+OT1
Fn126				-OT5	-OT4	-OT3	-OT2	-OT1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA					LMS	NAL	

[Input type] Setting input
[Data type] Bit path

#1 NAL If the tool enters the inhibition area of stored stroke limit 1 during manual operation:
0: An alarm is issued and the tool is stopped.
1: An alarm is not issued, the stroke limit reach signal is output to the PMC, and the tool is stopped.

> **NOTE**
> When the tool enters the inhibition area of stored stroke limit 1 due to the move command issued during automatic operation, even if this parameter is set to 1, an alarm is issued and the tool is stopped. Even in this case, the stroke limit reach signal is output to the PMC.

#2 LMS The EXLM signal for switching stored stroke check
0: Disabled
1: Enabled
When bit 0 (DLM) of parameter No. 1301 is set to 1, the stored stroke check 1 switch signal EXLM <G007.6> 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 (2-path)), 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		OF1				DLM

[Input type] Setting input
[Data type] Bit path

#0 DLM The stored stroke limit switching signals +EXLx and -EXLx for each axial direction are:
0: Disabled.
1: Enabled.
When this parameter is set to 1, the stored stroke check 1 switch signal EXLM <G007.6> is made invalid.

#4 OF1 If the tool is moved into the range allowed on the axis after an alarm is raised by stored stroke check 1,

0: The alarm is not canceled before a reset is made.
1: The OT alarm is immediately canceled.

NOTE
In the cases below, the automatic release function is disabled. To release an alarm, a reset operation is required.
1 When a setting is made to issue an alarm before a stored stroke limit is exceeded (bit 7 (BFA) of parameter No. 1300 is set to 1)
2 When an another overtravel alarm (such as stored stroke check 2, stored stroke check 3, and interference check) is already issued

#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 + and - directions 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 parameter No. 1320 and No. 1321 is a prohibited area.

1326	Coordinate value II of stored stroke check 1 in the negative direction on each axis

1327	Coordinate value II 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 + and - directions in the machine coordinate system.
When the stored stroke check switch signal EXLM is set to 1, or the stored stroke check switch signals for each axis direction +EXLx and -EXLx are set to 1, parameter No. 1326 and No. 1327 are used for stroke check instead of parameter No.1320 and No. 1321.

NOTE
1 Specify diameter values for any axes for which diameter programming is specified.
2 The area outside the area set by parameter No. 1326 and No. 1327 is a prohibited area.
3 The EXLM signal is valid only when bit 2 (LMS) of parameter No. 1300 is set to 1.
4 The +EXLx signal is valid only when bit 0 (DLM) of parameter No. 1301 is set to 1.

Alarm and message

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.

Caution

⚠ 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.

Note

NOTE
1 Bit 7 (BFA) of parameter No. 1300 can be used to specify whether to stop the tool before it enters the forbidden area or when it enters the area.
2 The forbidden area must be specified with correct positional relationships. An incorrect area setting results in stroke infinity.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Stroke check

2.3.3 Stored Stroke Check 2, 3

Overview

For stored stroke check 2, the outside or inside of the area specified by parameters or a program is defined as the forbidden area. As a limit position, specify a distance from the origin of the machine coordinate system. This function is enabled after manual reference position return is performed at power-on. When the limits are specified in a program, they can be set for the X-, Y-, and Z-axes. For this reason, the forbidden area can be changed according to the workpiece. Whether to define the inside or outside of the specified area as the forbidden area is determined by setting the corresponding parameter.

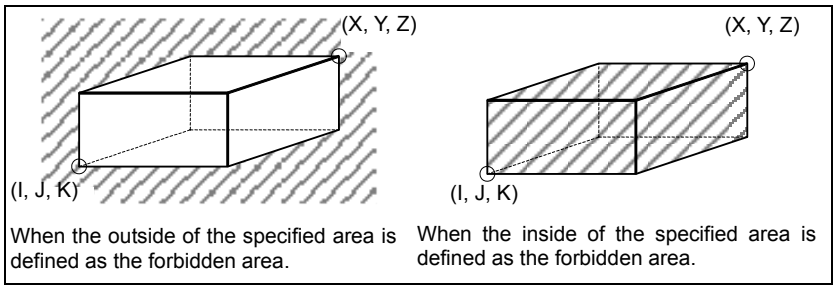


Fig. 2.3.3 (a) Stored stroke check 2

For stored stroke check 3, the inside of the area set by parameters is defined as the forbidden area.

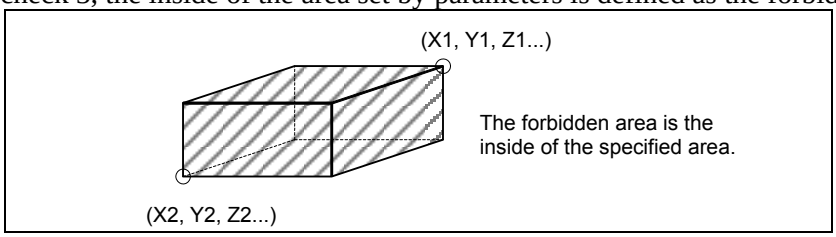


Fig. 2.3.3 (b) Stored stroke check 3

NOTE
To enable stored stroke checks 2 and 3, set 0 in bit 1 (BAR) of parameter No. 8134. In this case, chuck/tail stock barrier (T series) cannot be used.

Explanation

Three areas which the tool cannot enter can be specified with stored stroke check 1, stored stroke check 2, and stored stroke check 3.

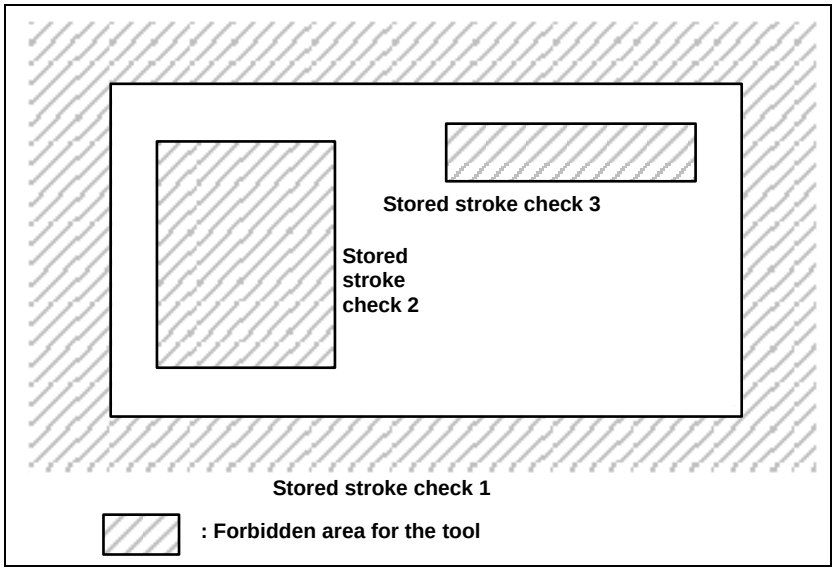


Fig. 2.3.3 (c) Stroke check

The following shows the areas which the tool cannot enter for each stored stroke check.

- Stroke check 1: Outside
- Stroke check 2: Outside or inside (switchable)
- 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.

- 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. Parameter OUT (No. 1300#0) 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 below creates or changes the forbidden area:

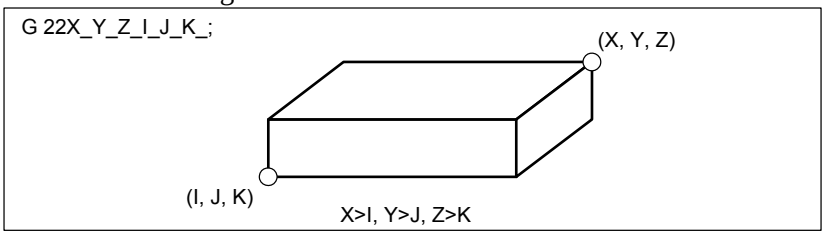


Fig. 2.3.3 (d) Creating or changing the forbidden area using a program

When setting the area by parameters, points A and B in the figure below must be set.

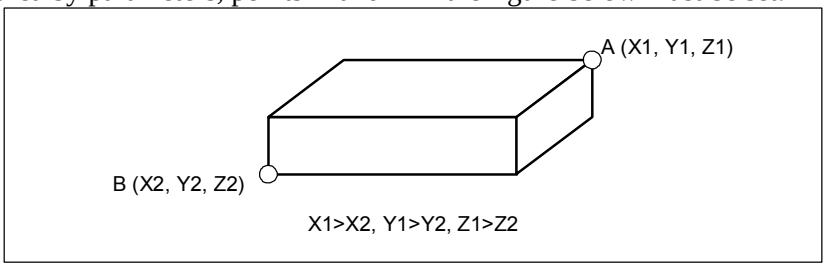


Fig. 2.3.3 (e) Creating or changing the forbidden area using a parameters

The values X1, Y1, Z1, X2, Y2, and Z2, which are set by parameters No. 1322 and No. 1323, must be specified by the distance from the machine coordinate system (machine increment). 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 increment).
Values set by a program are then converted in the machine increment and the values are set as the parameters.

- Format

Stored stroke check 2 on

G22 X__ Y__ Z__ I__ J__ K__ ;

X, Y, Z: Coordinate of stored stroke check 2 in the positive direction on each axis
I, J, K: Coordinate of stored stroke check 2 in the negative direction on each axis

Use addresses X, Y, and Z or addresses I, J, and K. Addresses X and I specify the forbidden area related to the X-axis of the basic three axes. Addresses Y and J specify the forbidden area related to the Y-axis of the basic three axes. Addresses Z and K specify the forbidden area related to the Z-axis of the basic three axes. When addresses are omitted, a stroke check is made based on the values set by parameters.

Stored stroke check 2 off

G23 ;

- Stored stroke check 3

Set the boundary with parameters No. 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 increment) in the machine coordinate system.

- Checkpoint for 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 Figs. 2.3.3 (f) and (g), 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.

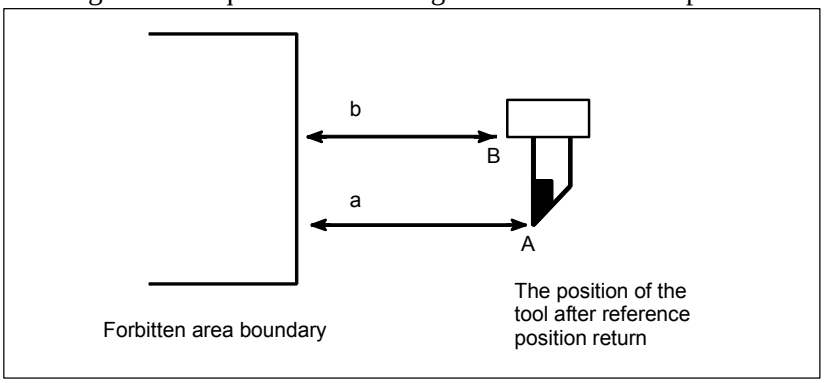


Fig. 2.3.3 (f) Setting the forbidden area (turning tool)

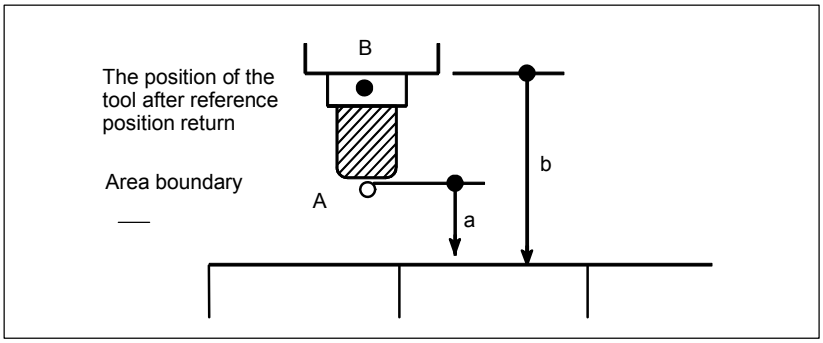


Fig. 2.3.3 (g) Setting the forbidden area (rotating tool)

- Forbidden area overlapping

Area can be set in piles.

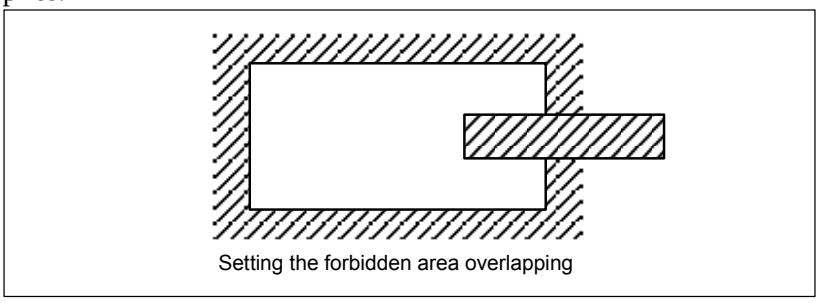


Fig. 2.3.3 (h) Setting the forbidden area overlapping

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

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.

Signals

Stroke check 3 release signal RLSOT3<Gn007.4>

[Classification] Input signal
[Function] Selects whether stored stroke check 3 is checked.
[Operation] When this signal is set to 1, does not check stored stroke check 3.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn007				RLSOT3				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA		RL3					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

#5 RL3 Stored stroke check 3 release signal RLSOT3 is
0: Disabled
1: Enabled

#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 (2-path)), 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
1310							OT3x	OT2x

[Input type] Setting input
[Data type] Bit axis

#0 OT2x Stored stroke check 2 for each axis is :
0: Disabled
1: Enabled

#1 OT3x Stored stroke check 3 for each axis is :
0: Disabled
1: Enabled

1322	Coordinate value I of stored stroke check 2 in the positive direction on each axis
1323	Coordinate value I 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 + and - directions 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 I of stored stroke check 3 in the positive direction on each axis
1325	Coordinate value I 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 + and - directions 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 parameter No. 1324 and No. 1325 is a prohibited area.

	#7	#6	#5	#4	#3	#2	#1	#0
3402	G23							

[Input type] Parameter input
[Data type] Bit path

#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
8134							BAR	

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#1 BAR Stored stroke limits 2 and 3 are:
0: Used.
1: Not Used.

NOTE
When the stored stroke limits 2 and 3 is selected, chuck and tail stock barrier function cannot be used.

That is, this parameter also specifies whether to use tail stock barrier function as shown below.

BAR Chuck and tail stock barrier function (T series) is:
0: Not Used.
1: Used.

Alarm and message

Number	Message	Description
OT0502	+ OVERTRAVEL (SOFT 2)	A movement in the positive direction exceeded stored stroke check 2. Alternatively, in the chuck and tail stock barrier (T series), an entry to the forbidden area was made during movement in the positive direction.
OT0503	- OVERTRAVEL (SOFT 2)	A movement in the negative direction exceeded stored stroke check 2. Alternatively, in the chuck and tail stock barrier (T series), an entry to the forbidden area was made during movement in the negative direction.
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.

Caution

⚠ CAUTION

1 If the two points for specifying a forbidden area are identical, all areas are handled as movable areas for check 2/3 which sets forbidden area internally by setting 0 in bit 0 (OUT) of parameter No.1300.
If the two points for specifying a forbidden area are identical, all areas are handled as forbidden areas for check 2 which sets forbidden area outside by setting 1 in bit 0 (OUT) of parameter No.1300.
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.

Note

NOTE

1 Bit 7 (BFA) of parameter No. 1300 can be used to specify whether to stop the tool before it enters the forbidden area or when it enters the area.
2 Because checking is made with higher precision than the precision of indicated coordinates, an OT alarm may be issued even when coordinates on a boundary is indicated.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Stroke check

2.3.4 Checking the Stored Stroke Limit during the Time from Power–on to the Reference Position Establishment

This function stores the machine coordinates present immediately before the power is turned off. Therefore, immediately after the power is turned on again, this function can restore the approximate machine coordinates and enables the function for checking the stored stroke limit during the time from power–on to the reference position establishment.
Even before the reference position is established by manual reference position return, the stored stroke limit check can be performed using approximate machine coordinates.

Absolute coordinates are set based on the machine coordinates.
Data such as workpiece offsets specified by G92, G52, and so on before the power is turned off, however, is not set.
If bit 3 (PPD) of parameter No. 3104 is set to 1, the relative position indication is preset.

The stored stroke limit check is enabled immediately after the power is turned on.
This stored stroke limit check is performed in the same way as the normal stored stroke limit check.

The reference position is not established until a reference position return takes place.
(Until then, reference position establishment signal ZRFn<Fn120> is 0.
Precision compensation functions such as pitch error compensation are not performed.)

NOTE

1 This function stores machine coordinates periodically while the CNC is operating.
This means that no machine coordinates are stored when the tool moves while the power is off.
If the power to the CNC is turned off while the tool is moving along an axis, correct machine coordinates cannot be stored.
To reflect movements made while the power is off, use an absolute position detector (absolute pulse coder).

2 Because this function stores machine coordinates by using software, load is imposed on the system.
For axes for which this function is not required, this function should not be set.

3 When an absolute position detector (abosolute pulse coder) is used for an axis, this function cannot be used for that axis.

4 If the tool is already beyond a stored stroke limit immediately after the power is turned on, an overtravel alarm (OT alarm) is issued.
In this case, release the alarm by moving the tool along the axis in the direction that does not cause overtravel.
If bit 6 (LZR) of parameter No. 1300 is set to 1, the function for checking the stored stroke limit during the time from power–on to the reference position establishment is disabled.

5 When the power is turned on while the [CAN] key and the [P] address key are held down, the stored stroke limit check immediately after power-on is disabled.
(When the MDI keys are provided as standard keys)
(When performing this operation, exercise special care.)
In this case, machine coordinates and absolute coordinates are set.
When the compact type MDI keys are used, the stored stroke limit immediately after power-on is disabled by holding down the [CAN] key and [O] address key at power-on.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1300		LZR						

[Input type] Setting input
[Data type] Bit path

#6 LZR When the stored stroke limit check immediately after power-on is enabled (bit 0 (DOT) of parameter No. 1311 is set to 1), the stored stroke check is:
0: Performed even before a manual reference position return is made.
1: Not performed until a manual reference position return is made.

	#7	#6	#5	#4	#3	#2	#1	#0
1311								DOTx

[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 DOTx Stored stroke limit check immediately after power-on is:
0: Disabled.
1: Enabled.

If the stored stroke limit check is enabled, the machine coordinate value present immediately before the power is turned off is stored.
The machine coordinate value is set immediately after the power is turned on.
Based on the machine coordinate value, absolute coordinate and relative coordinate values are set.

NOTE
Because this function uses software to store machine coordinates, the function puts an extra load on the system. So, this function should not be set for axes that do not require this function. The amount of a movement made while the power is off is not reflected in machine coordinates immediately after the power is turned on.

2.3.5 Stroke Limit External Setting (M Series)

M

Overview

When a tool is changed, the tool tip is aligned with the end of the limit area and signals are input. This operation sets the machine position (machine coordinates) at that time as the limit position in stored stroke check parameters. A setting signal is provided for each direction of each axis.

Signal

Stroke limit external setting signal +LM1 to +LM5 <Gn110.0 to Gn110.4>,-LM1 to -LM5 <Gn112.0 to Gn112.4>

[Classification] Input signal
[Function] Changes the values of the parameters for stroke check 1 (Nos. 1320 and 1321).
[Operation] When these signals are set to 1, the control unit changes the values of the parameters (Nos. 1320 and 1321) using the machine coordinates when the signals are input as the stroke check values.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn110				+LM5	+LM4	+LM3	+LM2	+LM1
Gn112				-LM5	-LM4	-LM3	-LM2	-LM1

2.3.6 Chuck and Tail Stock Barrier (T Series)

T

Overview

The chuck/tail stock barrier function prevents damage to the machine by checking whether the tool tip interferes with either the chuck or tail stock.

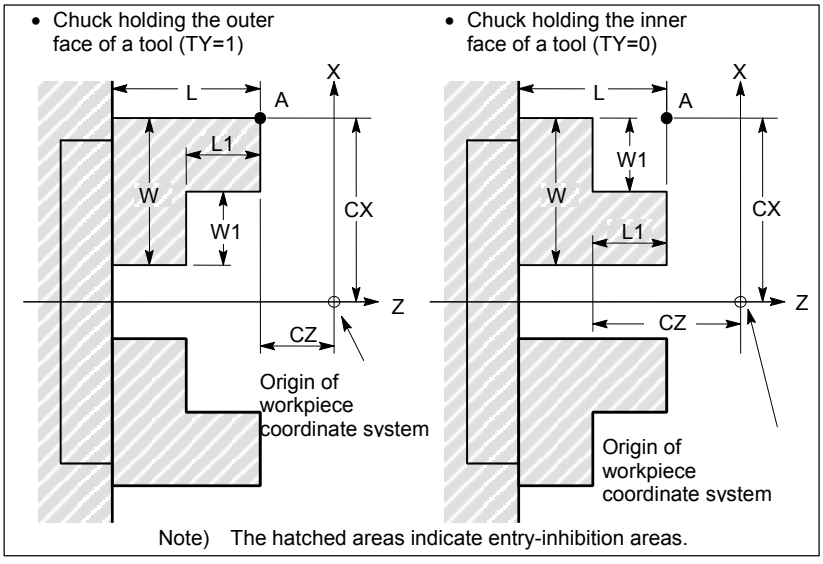
Specify an area into which the tool may not enter (entry-prohibition area). This is done using the special setting screen, according to the shapes of the chuck and tail stock. If the tool tip should enter the set area during a machining operation, this function stops the tool and outputs an alarm message.

The tool can be removed from the prohibited area only by retracting it in the direction from which the tool entered the area.

NOTE
1 To use chuck/tail stock barrier, set 1 in bit 1 (BAR) of parameter No. 8134.
2 When chuck/tail stock barrier is enabled (1 is set in bit 1 (BAR) of parameter No. 8134), stored stroke checks 2 and 3 cannot be used.

Explanation

- Setting the forbidden area for a chuck



Symbol	Description	Parameter
TY	Chuck-shape selection (0: Holding the inner face of a tool, 1: Holding the outer face of a tool)	No.1330
CX	Chuck position (along X-axis)	No.1335
CZ	Chuck position (along Z-axis)	No.1336
L	Length of chuck jaws	No.1331
W	Depth of chuck jaws (radius)	No.1332
L1	Holding length of chuck jaws	No.1333
W1	Holding depth of chuck jaws (radius)	No.1334

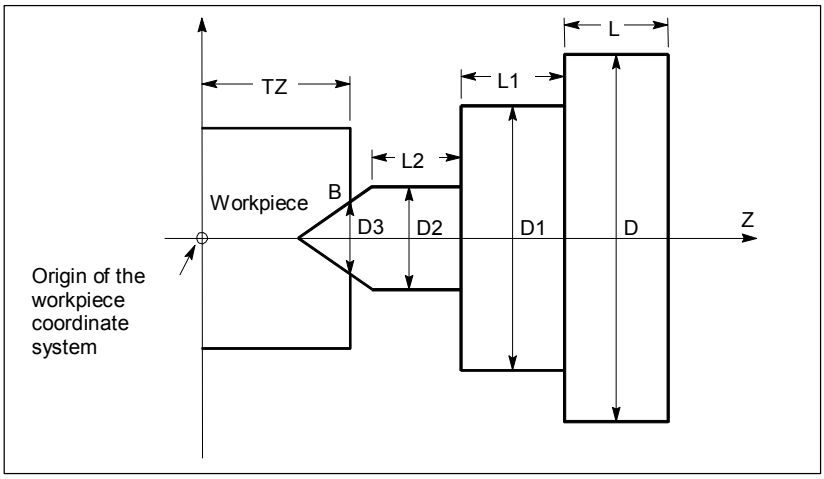
TY : Selects a chuck type, based on its shape. Specifying 0 selects a chuck that holds the inner face of a tool. Specifying 1 selects a chuck that holds the outer face of a tool. A chuck is assumed to be symmetrical about its Z-axis.

CX, CZ : Specify the coordinates of a chuck position, point A, in the workpiece coordinate system. These coordinates are not the same as those in the machine coordinate system.

L, L1 : Defines the length of chuck jaws.
W, W1 : Defines the width of chuck jaws

NOTE
1 For CX and CZ, whether diameter programming or radius programming is used for the axis determines the programming system. When diameter programming is used for the axis, use diameter programming to enter data for the axis.
2 Always specify W and W1 in radius. When radius programming is used for the Z-axis, specify L and L1 in radius.

- Setting the forbidden area for a tail stock



Symbol	Description	Parameter
TZ	Tailstock position (along the Z-axis)	No.1348
L	Tailstock length	No.1341
D	Tailstock diameter	No.1342
L1	Tailstock length (1)	No.1343
D1	Tailstock diameter (1)	No.1344
L2	Tailstock length (2)	No.1345
D2	Tailstock diameter (2)	No.1346
D3	Tailstock diameter (3)	No.1347

TZ : Specifies the Z coordinate of the chuck position, point B, in the workpiece coordinate system. These coordinates are not the same as those in the machine coordinate system. A tail stock is assumed to be symmetrical about its Z-axis.
L, L1, L2 : Defines the tail stock length.
D, D1, D2, D3 : Defines the tail stock diameter.

NOTE
1 For TZ, whether diameter programming or radius programming is used for the Z-axis determines the programming system.
2 Always specify D, D1, D2, and D3 in diameter programming. When radius programming is used for the Z-axis, specify L, L1, and L2 in radius.

Signal

Tailstock barrier select signal *TSB<Gn060.7>

[Classification] Input signal
[Function] Enables or disables the tail stock barrier.

[Operation] When this signal is set to 1, the control unit operates as follows:
Disables the tail stock barrier, even when the G22 command (stored stroke check on) is specified in the program.

G code	*TSB	Tail stock barrier	Chuck barrier (reference)
G22	0	Enabled	Enabled
	1	Disabled	Enabled
G23	0	Disabled	Disabled
	1	Disabled	Disabled

When the G23 command (stored stroke check off) is specified, the tail stock barrier is disabled regardless of the *TSB signal. When the G22 command (stored stroke check on) is specified, the tail stock can be disabled by setting the signal to 1.
This signal is used to select whether the tail stock area is a prohibited area.
It is used whenever M commands are applied, resulting in the tail stock being attached to the workpiece or detached from the workpiece while the workpiece is being machined.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn060	*TSB							

Limitation

- Correct setting of an entry-inhibition area

If an entry-inhibition area is incorrectly set, it may not be possible to make the area effective. Avoid making the following settings:

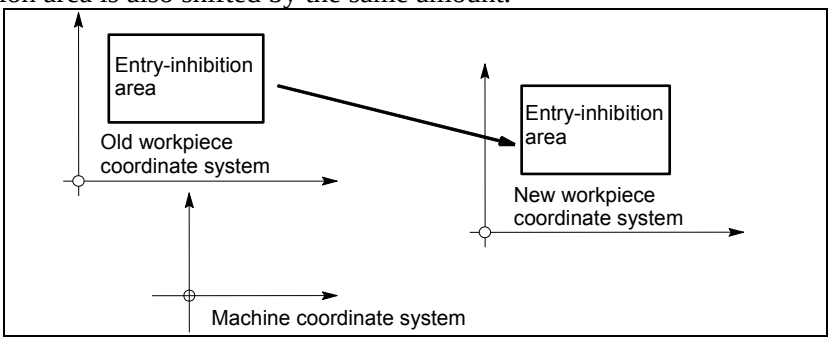
- L < L1 or W < W1 in the chuck-shape settings.
- D2 < D3 in the tail stock-shape settings.
- A chuck setting overlapping that of the tail stock.

- Retraction from the entry-inhibition area

If the tool enters the entry-inhibition area and an alarm is issued, switch to manual mode, retract the tool manually, then reset the system to release the alarm. In manual mode, the tool can be moved only in the opposite direction to that in which the tool entered the area.
The tool cannot be moved in the same direction (further into the area) as it was traveling when the tool entered the area.
When the entry-inhibition areas for the chuck and tail stock are enabled, and the tool is already positioned within those areas, an alarm is issued when the tool moves.
When the tool cannot be retracted, change the setting of the entry-inhibition areas, such that the tool is outside the areas, reset the system to release the alarm, then retract the tool. Finally, reinstall the original settings.

- Coordinate system

An entry-inhibition area is defined using the workpiece coordinate system. Note the following.
<1> When the workpiece coordinate system is shifted by means of a command or operation, the entry-inhibition area is also shifted by the same amount.



Use of the following commands and operations will shift the workpiece coordinate system.
Commands:
G54 to G59, G52, G50 (G92 in G code system B or C)
Operations:
Manual handle interrupt, change in offset relative to the workpiece reference point, change in tool offset (tool geometry compensation), operation with machine lock, manual operation with machine absolute signal off

<2> When the tool enters an entry-inhibition area during automatic operation, set the manual absolute signal, *ABSM<Gn006.2>, to 0 (on), then manually retract the tool from the area. If this signal is 1, the distance the tool moves in manual operation is not counted in the tool coordinates in the workpiece coordinate system. This results in the state where the tool can never be retracted from the entry-inhibition area.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1006					DIAx			

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#3 DIAx The move command for each axis is based on:
0: Radius specification
1: Diameter specification

NOTE
For the FS0*i*-C, one of the following changes is required besides setting bit 3 (DIAx) of parameter No. 1006 so that the axis based on diameter specification achieves the specified amount of movement.
- Halve the command multiplication (the detection unit is not changed).
- Halve the detection unit and double the flexible feed gear (DMR).

For the FS0*i*-D, only if bit 3 (DIAx) of parameter No. 1006 is set, the CNC halves the specified pulse. Accordingly, the above changes are not required (when the detection unit is not changed). To halve the detection unit, double both CMR and DMR.

	#7	#6	#5	#4	#3	#2	#1	#0
8134							BAR	

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#1 BAR Chuck and tail stock barrier function is:
0: Not Used.
1: Used.

> **NOTE**
> When the chuck and tail stock barrier function is selected, stored stroke limits 2 and 3 cannot be used.

- Profile of a chuck

1330	Profile of a chuck

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to 1
Select a chuck figure.
0 : Chuck which holds a workpiece on the inner surface
1 : Chuck which holds a workpiece on the outer surface

1331	Dimensions of the claw of a chuck (L)

[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 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 length (L) of the claw of the chuck.

> **NOTE**
> Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1332	Dimensions of the claw of a chuck (W)

[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 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 width (W) of the claw of the chuck.

> **NOTE**
> Specify this parameter by using a radius value at all times.

1333	Dimensions of the part of a claw at which a workpiece is held (L1)

[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 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 length (L1) of the claw of the chuck.

NOTE
Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1334	Dimensions of the part of a claw at which a workpiece is held (W1)

[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 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 width (W1) of the claw of the chuck.

NOTE
Specify this parameter by using a radius value at all times.

1335	X coordinate of a chuck (CX)

[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 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 chuck position (X coordinate) in the workpiece coordinate system.

NOTE
Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1336	ZX coordinate of a chuck (CZ)

[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 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 chuck position (Z coordinate) in the workpiece coordinate system.

NOTE
Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

- Profile of a tail stock

1341	Length of a tail stock (L)

[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 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 length (L) of the tail stock.

NOTE
Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1342	Diameter of a tail stock (D)

[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 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 diameter (D) of the tail stock.

NOTE
Specify this parameter by using a diameter value at all times.

1343	Length of a tail stock (L1)

[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 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 length (L1) of the tail stock.

NOTE
Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1344	Diameter of a tail stock (D1)

[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 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 diameter (D1) of the tail stock.

NOTE
Specify this parameter by using a diameter value at all times.

1345	Length of a tail stock (L2)

[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 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 length (L2) of the tail stock.

NOTE
Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

1346	Diameter of a tail stock (D2)

[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 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 diameter (D2) of the tail stock.

NOTE
Specify this parameter by using a diameter value at all times.

1347	Diameter of the hole of a tail stock (D3)

[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 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 diameter (D3) of the tail stock.

NOTE
Specify this parameter by using a diameter value at all times.

1348	Z coordinate of a tail stock (TZ)

[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 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 tail stock position (Z coordinate) in the workpiece coordinate system.

NOTE
Whether to specify this parameter by using a diameter value or radius value depends on whether the corresponding axis is based on diameter specification or radius specification.

	#7	#6	#5	#4	#3	#2	#1	#0
3402	G23							

[Input type] Parameter input
[Data type] Bit path

#7 G23 When the power is turned on
0: G22 mode (stored stroke check on)
1: G23 mode (stored stroke check off)

Alarm and message

Number	Message	Description
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.

Warning

⚠ WARNING

1 When the stored stroke check 2 or 3 and chuck/tail stock barrier option are used at the same time, the chuck/tail stock barrier (T series) is valid and stored stroke check 2 or 3 is ignored. To use stored stroke checks 2 and 3, set 0 in bit 1 (BAR) of parameter No. 8134, and disable chuck/tail stock barrier.

2 If an alarm is issued, the tool stops before it enters the entry-forbidden area when bit 7 (BFA) of parameter No. 1300 is set to 1.
When bit 7 (BFA) of parameter No. 1300 is set to 0, an actual stop position can be in the forbidden area beyond the specified profile because the CNC and machine system stop with some delay in time. For this reason, for safety, set profile data a little larger than the actual profile. The extra distance, L, required for this purpose is calculated from a rapid traverse rate as follows:
L = (rapid-traverse-rate) $\times$ 1/7500
For example, when a rapid traverse rate of 15 m/min is used, set profile data 2 mm larger than the actual profile.
The profile of a chuck or tail stock can also be set using parameters Nos. 1330 to 1348.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (Lathe system) (B-64304EN-1)	Check/tail stock barrier

2.3.7 Stroke Limit Check Before Move

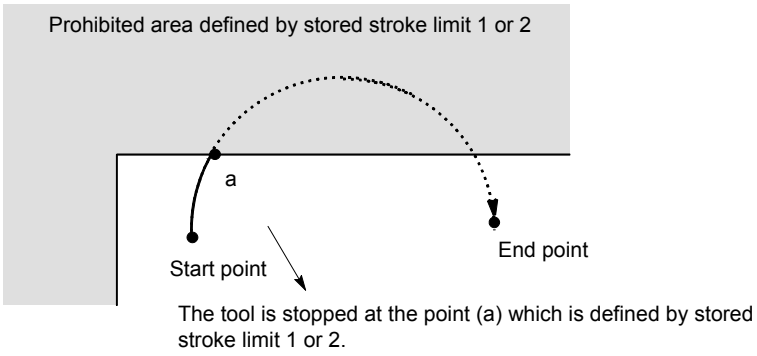
Overview

In automatic operation, before executing the move command by a given block, the position of the end point is determined. It is calculated from the current position of the machine and from the specified amount of travel. It is also determined whether or not the tool will enter the prohibited area defined by stored stroke check 1, 2 or 3. It it is determined that the tool will enter the prohibited area defined by a stored stroke check, the tool is stopped immediately once this block starts execution, and an alarm is displayed.

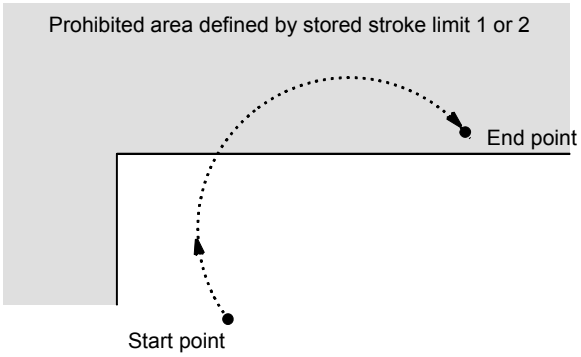
⚠ WARNING

Only the coordinates of the end point, reached as a result of traversing the distance specified in each block, are checked against the prohibited area. The coordinates along the path are not checked. However, if the tool enters the prohibited area defined by stored stroke check 1, 2, or 3, an alarm is issued at that point along the path.

Example 1)



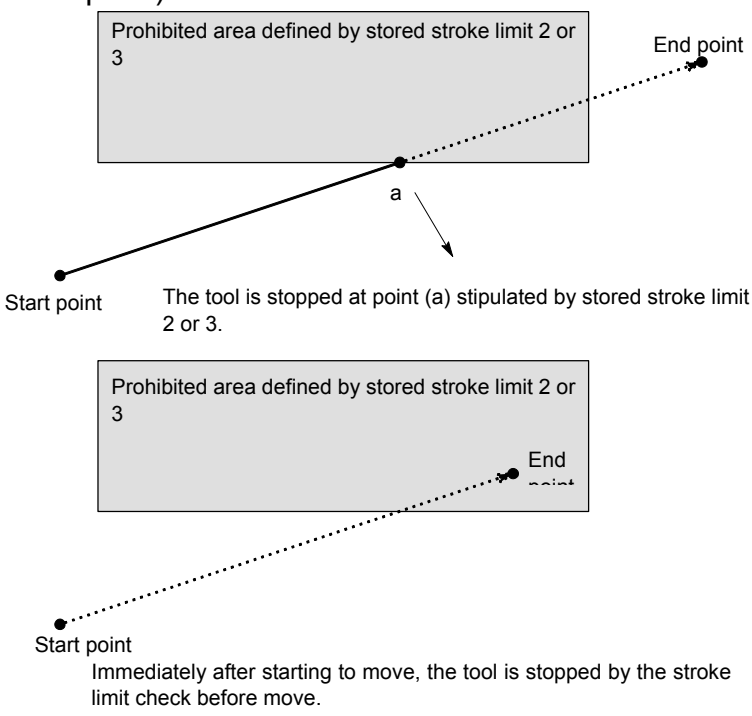
The tool is stopped at the point (a) which is defined by stored stroke limit 1 or 2.



Immediately after starting to move, the tool is stopped by the stroke limit check before move.

⚠ WARNING

Example 2)



Explanations

When a stroke limit check before move is performed, bit 2 (NPC) of parameter No. 1301 is used to determine whether to check the moves performed by G31 (skip) or G37 (automatic tool length measurement).

Limitations

- 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 limit 2 is disabled (G23 mode), no check is made to determine whether the tool enters the prohibited area defined by stored stroke limit 2.

- Program restart

When a program is restarted, an alarm is issued if the restart position is within a prohibited 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 though the end point following a manual intervention is within a prohibited 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 a prohibited 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.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1301	PLC					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 (M series) or automatic tool compensation (T series)) blocks is:
0: Checked
1: Not checked

#7 PLC Stroke limit check before movement is:
0: Not performed
1: Performed

Alarm and message

Number	Message	Contents
OT0510	+ OVERTRAVEL (PRE-CHECK)	The stroke limit check made prior to performing movement reveals that the end point of a block is located within the stroke limit prohibited area in the positive direction. Correct the program or redefine the prohibited area.
OT0511	- OVERTRAVEL (PRE-CHECK)	The stroke limit check made prior to performing movement reveals that the end point of a block is located within the stroke limit prohibited area in the negative direction. Correct the program or redefine the prohibited area.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Stroke limit check before move

2.4 ALARM SIGNALS

When an alarm is triggered in the CNC, the alarm is displayed on the screen, and the alarm signal is set to 1.
If the voltage level of the memory backup battery falls to below a specified level while the CNC is turned off, the battery alarm signal is set to 1.

Signal

Alarm signal AL<Fn001.0>

[Classification] Output signal
[Function] The alarm signal reports that the CNC is in an alarm state.

The following are the alarms that may be issued:

(a) TH alarm
(b) TV alarm
(c) P/S alarm
(d) Overtravel alarm
(e) Overheat alarm
(f) Servo alarm

[Output cond.] The alarm signal is set to 1 when:

- The CNC is placed in the alarm state.

The alarm signal is set to 0 when:

- The alarm has been released by resetting the CNC.

Battery alarm signal BAL<Fn001.2>

[Classification] Output signal

[Function] The battery alarm signal indicates that the voltage of the battery for the memory has fallen to below a specified level while the CNC is off. In general, this signal is used to turn on an LED to notify the operator.

[Output cond.] The signal is set to 1 when:

- The battery voltage has fallen to below the specified level.

The signal is set to 0 when:

- The battery voltage has risen to the specified level or higher.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn001						BAL		AL

2.5 START LOCK / INTERLOCK

These signals disable machine movement along axes. When any of these signals is activated during movement, tool movement along the affected axis (or axes) is decelerated, then stopped.

Signal

Start lock signal STLK<Gn007.1>

[Classification] Input signal

[Function] This signal disables movement along axes during automatic operation (memory operation, DNC operation, or MDI operation).

[Operation] When the STLK signal is set to 1, a movement along an axis decelerates and stops. In this case, the automatic operation mode is maintained (with the STL signal set to 1 and the SPL signal set to 0) while the movement is stopped. Therefore, when blocks containing only M, S, T, and B (second auxiliary functions) commands with no axis move commands are specified successively, the M, S, T, and B functions are executed one after another until a block specifying an axis move command is encountered. When an axis move command and M, S, T, and B functions are both specified, only the M, S, T, and B functions are sent, and a stop occurs in the automatic operation mode. Setting the STLK signal to 0 restarts the operation. (Figs. 2.5(a), (b)).

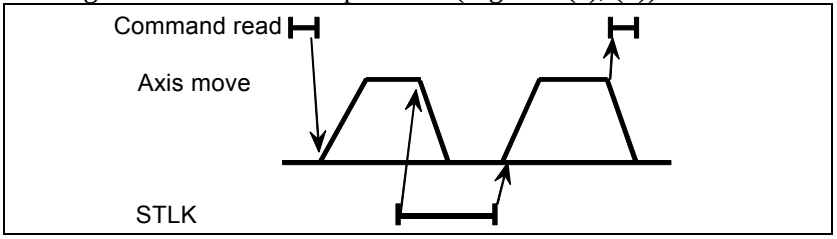


Fig.2.5 (a) Interlock when only axis move commands are specified

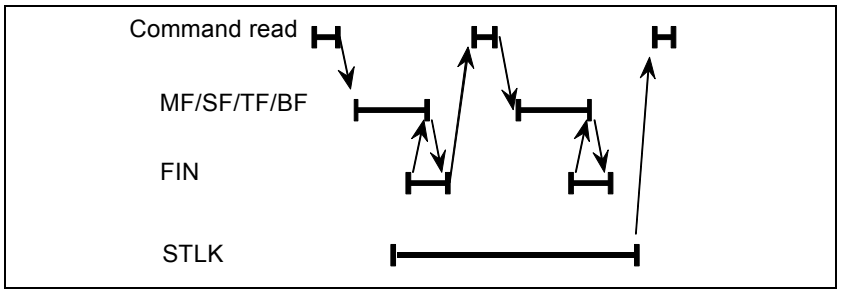


Fig.2.5 (b) Interlock when a block containing auxiliary functions only is specified

All axes interlock signal *IT<Gn008.0>

[Classification] Input signal

[Function] This signal is used to inhibit the machine from moving, and is effective regardless of the selected mode.

[Operation] When the *IT signal is set to "0", the axis movement is decelerated and stopped. In this case, the automatic operation mode is maintained (with the STL signal set to 1 and the SPL signal set to 0) while the movement is stopped. Therefore, when blocks containing only M, S, T, and B (second auxiliary functions) commands with no axis move commands are specified successively, the M, S, T, and B functions are executed one after another until a block specifying an axis move command is encountered. When an axis move command and M, S, T, and B functions are both specified, only the M, S, T, and B functions are sent, and a stop occurs in the automatic operation mode. Setting the *IT signal to 1 restarts the operation. (Figs. 2.5(c), (d)).

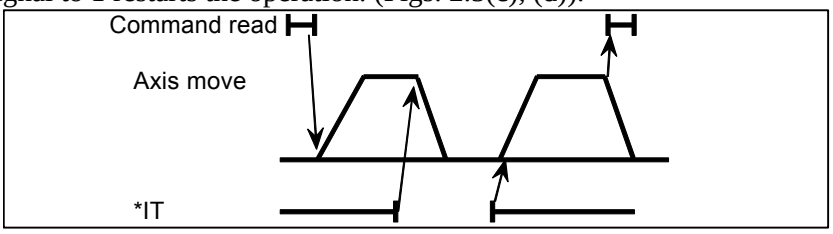


Fig. 2.5 (c) Block containing axis move commands only (manual and automatic operation)

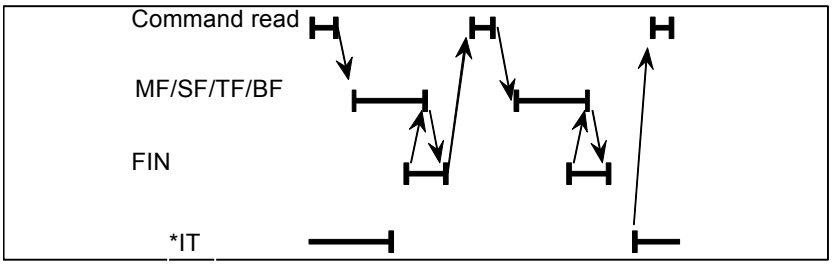


Fig. 2.5 (d) Block containing auxiliary functions only (automatic operation)

NOTE

The overtravel distance of the motor after turning *IT to "0" is represented by the following formula.

Where

$Qmax = Fm \times 1/60 \times (A/1000 + Tc/1000 + Ts/1000)$

Qmax : Overtravel quantity (mm or inch)
Fm : Feedrate (mm/min or inch/min)
Tc : Cutting time constant (ms)
Ts : Servo time constant (Ts = 33ms normally)
A : Processing time of CNC
A = 50ms

When acceleration/deceleration before read-ahead interpolation is used, the distance required for deceleration using acceleration/deceleration before interpolation is added to the above formula.

Interlock signal for each axis *IT1 to *IT5 <Gn130.0 to Gn130.4>

[Classification] Input signal

[Function] These signals disable feed along axes on an axis-by-axis basis.
A separate interlock signal is provided for each controlled axis. The number at the end of each signal name denotes the number of the corresponding controlled axis.

*IT x
x : 1 ..... Interlock for the first axis
2 ..... Interlock for the second axis
3 ..... Interlock for the third axis
: :

[Operation]

a) In manual operation
The movement of an interlocked axis is inhibited, but the other axes are movable. If an axis is interlocked during movement, it stops after being decelerated, and it starts moving again when it is released from interlock.

b) In automatic operation (MEM, RMT, or MDI mode)
If an axis is interlocked while its movement is being commanded (the move amount is not 0, inclusive of the tool offset), movement in all axes is prevented.
If a moving axis is interlocked, all axes stop moving after being decelerated, and they start moving again when interlock is released.
This function is also effective during dry run.

Interlock signal for each axis direction
(M series) +MIT1 to +MIT5<Gn132.0 - Gn132.4>,-MIT1 to -MIT5<Gn134.0 - Gn134.4>
(T series) +MIT1<X004.2>,-MIT1<X004.3>,+MIT2<X004.4>,-MIT2<X004.5>

[Classification] Input signal

[Function] An axis direction interlock can be applied for each axis.

[Operation] When an interlock signal for each axis direction is set to 1, the CNC applies an interlock to the specified axis direction only. During automatic operation, however, movements along all axes are stopped.

NOTE
1 With the T series, if bit 4 (DAU) of parameter No. 3003 is set to 0, an axis direction interlock is applied only during manual operation. To apply axis direction interlock also during automatic operation, set bit 4 (DAU) of parameter No. 3003 to 1.
2 With the T series, if bit 0 (GSC) of parameter No. 5009 is set to 1, the same signal addresses as the M series (<Gn132> and <Gn134>) can be set.

Block start interlock signal *BSL<Gn008.3>

[Classification] Input signal

[Function] This signal disables the start of the next block during automatic operation

[Operation] While this signal is 0, the execution of the next block during automatic operation is not started. This signal does not affect a block that has already started, and that block is continuously executed until its end. This signal does not halt automatic operation. The command in the next block is ready for execution as a valid command, so execution restarts as soon as the signal is set to 1.

NOTE
When blocks for cycle operation are internally created by a canned cycle and so on, only the first block is generally interlocked by this signal. The intermediate blocks are executed continuously even if this signal is set to 0.

Cutting block start interlock signal *CSL<Gn008.1>

[Classification] Input signal

[Function] This signal disables the start of blocks specifying move commands other than positioning during automatic operation.

[Operation] While this signal is 0, the execution of blocks specifying movement other than positioning during automatic operation is not started. This signal does not affect a block that has already started, and that block is continuously executed until its end. This signal does not halt automatic operation. The command in the next block is ready for execution as a valid command, so execution restarts as soon as the signal is set to 1.

[Usage] When the spindle has been specified, or when the spindle speed has been changed, this signal can be held 0 until a target spindle speed is achieved. Then, the next cutting block can be executed at the target spindle speed.

NOTE
This signal is effective for any blocks including blocks for cycle operation internally created by a canned cycle and so on.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn007							STLK	
Gn008					*BSL		*CSL	*IT
Gn130				*IT5	*IT4	*IT3	*IT2	*IT1
Gn132				+MIT5	+MIT4	+MIT3	+MIT2	+MIT1
Gn134				-MIT5	-MIT4	-MIT3	-MIT2	-MIT1
X004			-MIT2	+MIT2	-MIT1	+MIT1		
X013			-MIT2#2	+MIT2#2	-MIT1#2	+MIT1#2		

NOTE

For the T series, the +MIT1, -MIT1, +MIT2, and -MIT2 signals are placed in X004 (path 1) and X013 (path 2) shown above when bit 2 (XSG) of parameter No. 3008 is 0. When bit 2 (XSG) of parameter No. 3008 is 1, they are placed in the X addresses set by parameter No. 3019. (The bit positions do not change.) If bit 0 (GSC) of parameter No. 5009 is set to 1, an input signal can be input from the PMC side (Gn132 (- direction), Gn134 (+ direction)). In this case, the bit positions are the same as those in the M series.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3003				DAU	DIT	ITX		ITL
					DIT	ITX		ITL

[Input type] Parameter input
[Data type] Bit path

#0 ITL Interlock signal for all axes
0: Enabled
1: Disabled

#2 ITX Interlock signals for each axis
0: Enabled
1: Disabled

#3 DIT The interlock signal for each axis direction is:
0: Valid.
1: Invalid.

#4 DAU When bit 3 (DIT) of parameter No. 3003 is set to 0, the interlock signal for each axis direction is:
0: Valid only in manual operation, and invalid in automatic operation.
1: Valid in either manual operation or automatic operation.

	#7	#6	#5	#4	#3	#2	#1	#0
3004							BCY	BSL

[Input type] Parameter input

[Data type] Bit path

#0 BSL The block start interlock signal (*BSL) and cutting block start interlock signal (*CSL) are:
0: Disabled.
1: Enabled.

#1 BCY When more than one operation is performed by one block command such as a canned cycle, the block start interlock signal (*BSL) is:
0: Checked only at the beginning of the first cycle.
1: Checked at the beginning of every cycle.

	#7	#6	#5	#4	#3	#2	#1	#0
5009								GSC

[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 GSC When the function for direct input of offset value measured B is used, an offset write input signal is input from:
0: Machine side
1: PMC side

NOTE
When interlock for each axis direction is enabled (bit 3 of parameter No. 3003 is set to 0), the input for the interlock for each axis direction can also be switched from the machine side to PMC side at the same time.

Note

NOTE
1 The interlock signal for each axis direction (T series) can be used regardless of whether the function for direct input of offset value measured B is provided or not.
2 When an interlock is being applied to an axis, "I" appears on the left side of the axis address of the axis on the coordinate position display screen.

2.6 MODE SELECTION

Overview

The mode selection signal is a code signal consisting of the three bits MD1, MD2, and MD4.
The following five modes can be selected.

- Memory edit (EDIT)
- Memory operation (MEM)
- Manual data input (MDI)
- Manual handle/incremental feed (HANDLE/INC)
- Manual continuous feed (JOG)

And in addition, DNC operation mode can be selected by combining the memory operation (MEM) mode setting and the DNCI signal. Manual reference position return mode can be selected by combining the manual continuous feed (JOG) mode setting and the ZRN signal.
The currently selected operation mode can be posted by outputting the operation mode check signal.

Signals

Signals for mode selection

Select the desired mode by setting signals for mode selection listed below according to the setting shown in Table 2.6 (a).

Mode selection signals MD1,MD2,MD4<Gn043.0 to .2>

DNC operation selection signal DNCI<Gn043.5>

Signal for selecting manual reference position return ZRN<Gn043.7>

Signals for mode check

Check the current mode by reading the mode check signal(s) according to the setting shown in Table 2.6 (a).

Signal for checking incremental feed selection MINC<Fn003.0>

Signal for checking manual handle feed selection MH<Fn003.1>

Signal for checking jog feed selection MJ<Fn003.2>

Signal for checking manual data input selection MMDI<Fn003.3>

Signal for checking DNC operation selection MRMT<Fn003.4>

Signal for checking memory operation selection MMEM<Fn003.5>

Signal for checking program edit selection MEDT<Fn003.6>

TEACH IN selection check signal MTCHIN<Fn003.7>

Signal for checking manual reference position return selection MREF<Fn004.5>

Table 2.6 (a) Mode selection signals and corresponding check signals
(A hyphen (-) indicates that the mode is not related to the status of the signal.)

Mode		Input signal					Output signal
		MD4	MD2	MD1	DNCI	ZRN	
Automatic operation	Manual data input, MDI operation (MDI)	0	0	0	-	-	MMDI
	Memory operation (MEM)	0	0	1	0	-	MMEM
	DNC operation (RMT)	0	0	1	1	-	MRMT
Memory edit (EDIT)		0	1	1	-	-	MEDT
Manual operation	Manual handle/ incremental feed (HANDLE/INC)	1	0	0	-	-	MH,MINC
	Jog feed (JOG)	1	0	1	-	0	MJ
	Manual reference position return (REF)	1	0	1	-	1	MREF
	TEACH IN JOG (TJOG)	1	1	0	-	-	MTCHIN,MJ
	TEACH IN HANDLE (THND)	1	1	1	-	-	MTCHIN,MH

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn043	ZRN		DNCI			MD4	MD2	MD1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn003	MTCHIN	MEDT	MMEM	MRMT	MMDI	MJ	MH	MINC
Fn004			MREF					

Note

Precautions on modes and mode switching

NOTE

1 In MDI mode, the STL signal turns to "0" and the CNC stops as soon as the commands entered via the MDI have been executed. But the SPL signal does not turn to "1". Therefore, another command can be entered from the manual data input unit under this condition.

2 Manual operation in jog feed mode
   (1) When bit 0 (JHD) of parameter No. 7100 is set to 0
   Only jog feed is possible.
   (2) When bit 0 (JHD) of parameter No. 7100 is set to 1
   Both jog feed and manual handle feed are possible, provided the manual handle feed function is installed.

3 Manual operation in manual handle/incremental feed mode.
   (1) When the manual handle feed function is not used
   Incremental feed is valid.
   (2) When the manual handle feed function is used and bit 0 (JHD) of parameter No. 7100 is set to 0
   Only manual handle feed is valid.
   (3) When the manual handle feed function is used and JHD is set to 1
   Incremental feed and manual handle feed are valid.

NOTE

4 When switching to manual data input mode is made during operation in memory operation mode, the CNC enters the automatic operation stop state after executing the command in the current block. Signal STL is then set to 0.
In this case, signal SPL is not set to 1 (Fig. 2.6 (a)). When switching to memory operation mode is made during operation in manual data input mode, the CNC enters memory operation mode after executing the currently executed command.
To perform memory operation at this time, the program for MDI operation must have been executed (Fig. 2.6 (b))

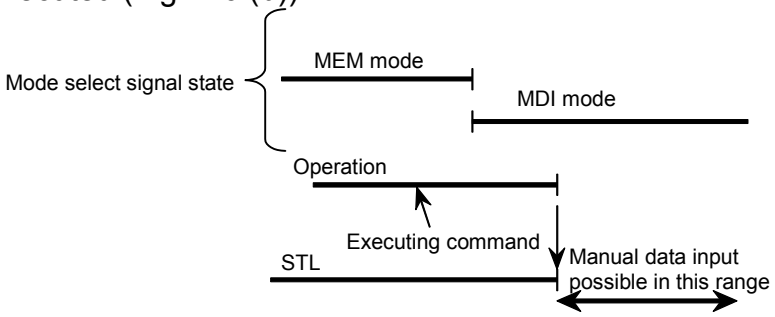


Fig. 2.6 (a)

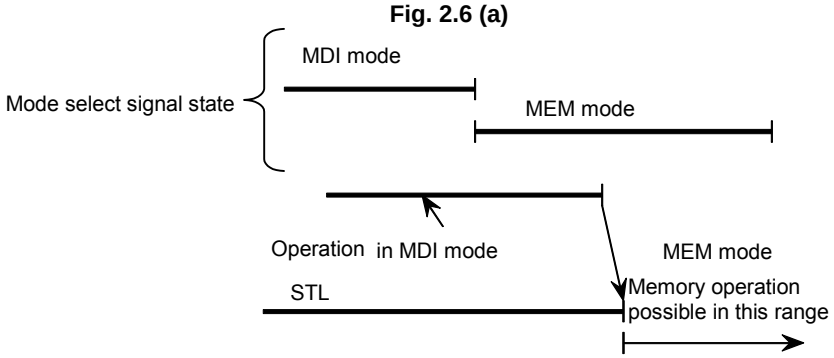


Fig. 2.6 (b)

NOTE

5 When the HANDLE/INC mode is selected while the CNC is operating in the MEM, RMT or MDI mode, the automatic or MDI operation stops, the STL signal turns to "0", the SPL signal simultaneously turns to "1", and the CNC enters the HANDLE/INC mode. Under these conditions, manual handle feed or incremental feed by axis direction select signal is permitted. Since the MEM, RMT or MDI mode commands are held, operation can be restarted by the cycle start signal and by selecting the MDI, RMT or MEM mode. However, if operation was stopped by switching to the HANDLE/INC mode during manual data input or during automatic operation, it can be restarted only by reactivating the mode in use before the operation was stopped (Fig. 2.6 (c)).

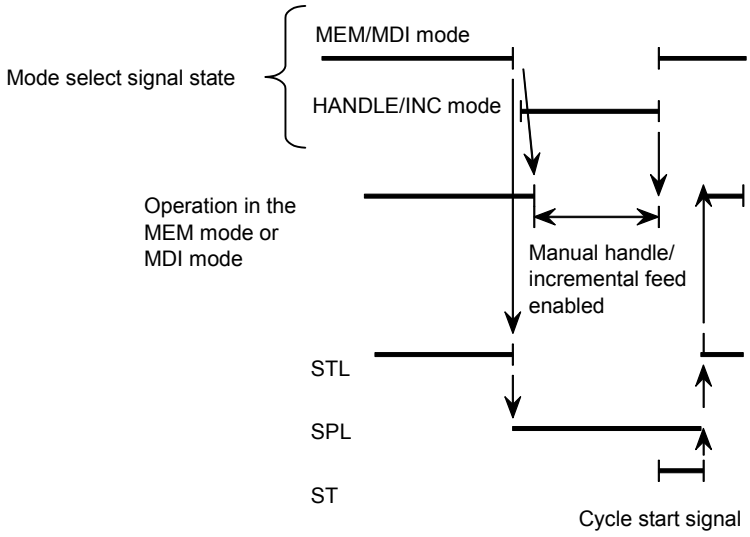


Fig. 2.6 (c)

NOTE

6 When the JOG mode is selected during MEM, RMT or MDI mode operation, operation stops, the STL signal turns to "0", the SPL signal simultaneously turns to "1", and the CNC enters the JOG mode.
Under these conditions, manual feed by feed axis direction select signal is permitted. Operation can be restarted by returning to the original mode, as described for HANDLE/INC mode (Fig. 2.6 (d)). When the mode is switched to the JOG mode during manual handle feed or during incremental feed operation, the CNC ignores the manual handle feed or incremental feed command and manual jog feed becomes effective. If a feed axis direction select signal turns to "1" before the JOG mode is selected, that signal is ignored. The feed axis select signal is selected by turning the necessary feed axis direction signal to "1" after turning all the feed axis direction select signals to "0" (Fig. 2.6 (e)).
It is possible to perform handle feed in JOG mode by activating parameter JHD (No.7100#0). For details, refer to Note 2.

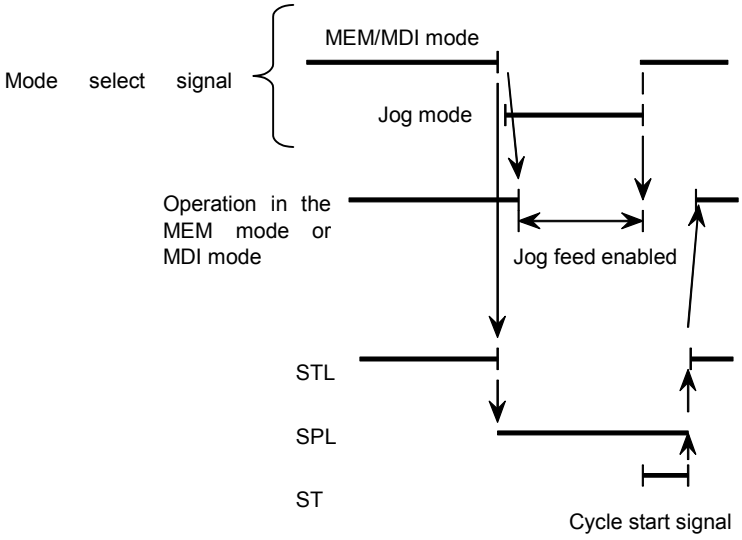


Fig. 2.6 (d)

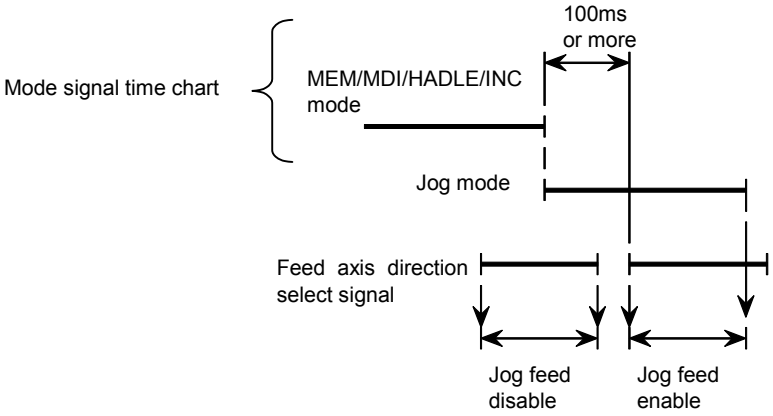


Fig. 2.6 (e)

NOTE

7 The mode switching operation is summarized in the time chart below (Fig. 2.6 (f)).

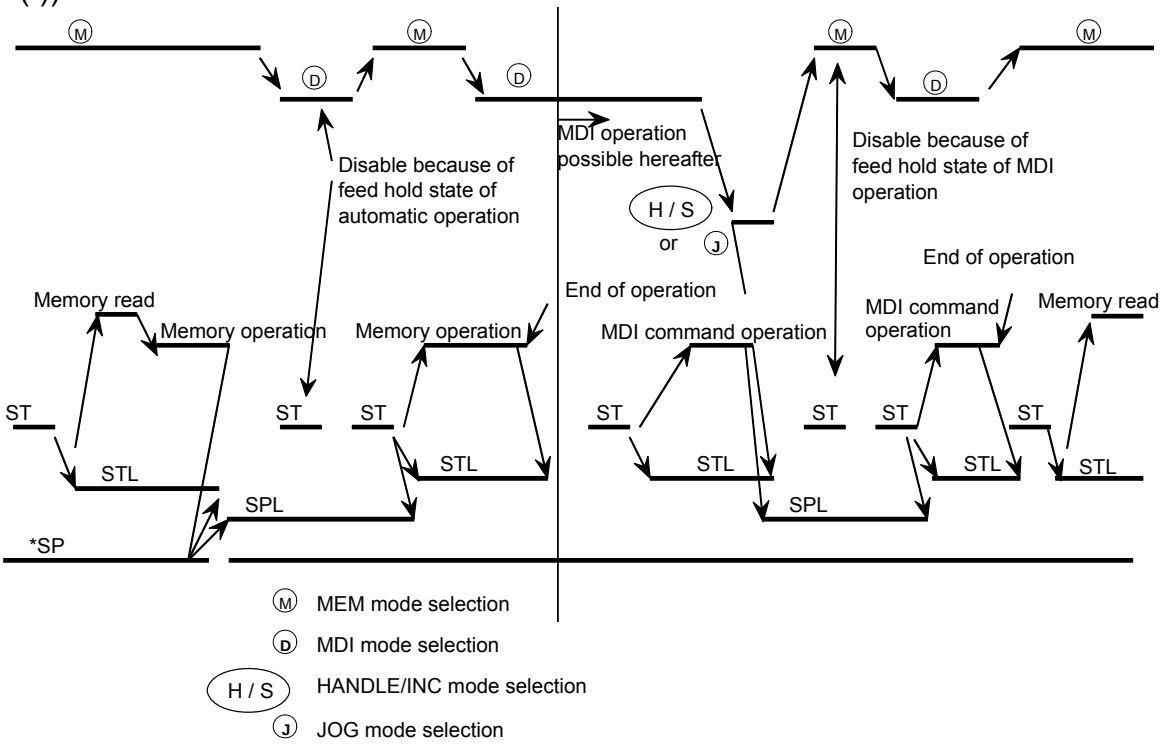


Fig. 2.6 (f) Mode signal time chart

8 Invalid mode selection signals
When mode selection signals MD4, MD2, and MD1 are set to 0, 1, and 0, respectively, the mode before the signals are changed is selected. When MD4, MD2, and MD1 have been set to 0, 1, and 0, respectively, since power-on, the MDI mode is selected.

9 Notes on output signals
1) When parameter JHD is set to 1:
In the jog feed mode, the output signals are set as follows:
MJ=1, MH=0
In the manual handle feed mode, the output signals are set as follows:
MH=1, MINC=0

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7100								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
0: Invalid
1: Valid

	When JHD:=0		When JHD:=1	
	JOG feed mode	Manual handle feed mode	JOG feed mode	Manual handle feed mode
JOG feed	○	×	○	×

	When JHD:=0		When JHD:=1	
	JOG feed mode	Manual handle feed mode	JOG feed mode	Manual handle feed mode
Manual handle feed	×	○	○	○
Incremental feed	×	×	×	○

2.7 STATUS OUTPUT SIGNAL

The table below lists the status output signals. They indicate the state of the CNC. See the sections listed in the table for details of each signal.

Signal name	Symbol	Reference section
Alarm signal	AL	Alarm signals
Battery alarm signal	BAL	Alarm signals
Reset signal	RST	Reset & rewind
Rewinding signal	RWD	Reset & rewind
Tapping signal	TAP	Canned cycle for drilling
Moving signals	MV1 to MV5	Axis moving status output
Moving direction signals	MVD1 to MVD5	Axis moving status output
In-position signals	INP1 to INP5	In-position check
Rapid traversing signal	RPDO	This section
Cutting feed signal	CUT	This section
Threading signal	THRD	Threading
Constant surface speed signal	CSS	Constant surface speed control
Inch input signal	INCH	Inch/metric switching

Signal

Rapid traversing signal RPDO<Fn002.1>

[Classification] Output signal

[Function] This signal indicates that a move command is being executed at rapid traverse.

[Output cond.] "1" indicates an axis starts moving after rapid traverse has been selected; "0" indicates that an axis starts moving after a feedrate other than rapid traverse has been selected. This holds true for both automatic and manual operation modes.

NOTE
1 Rapid traverse in automatic operation includes all rapid traverses in canned cycle positioning, automatic reference position return, etc., as well as the move command G00. Rapid traverse in manual operation also includes rapid traverse in reference position return.
2 Once rapid traverse has been selected, this signal remains "1", including during a stop, until another feedrate has been selected and movement has been started.

Cutting feed signal CUT<Fn002.6>

[Classification] Output signal

[Function] Signals that cutting feed is being performed by automatic operation.

[Output cond.] This signal is set to 1 in the following case:

- When cutting feed is being performed by automatic operation (cutting feed for linear interpolation, circular interpolation, helical interpolation, thread cutting, skip cutting, or cutting in canned cycle)

> ⚠ **CAUTION**
> This signal is not set to "1" in the feed hold state.

> **NOTE**
> This signal is set to "1" even when the feedrate override is 0%, and even during interlock.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn002		CUT					RPDO	

2.8 VRDY OFF ALARM IGNORE SIGNAL

Overview

The German VDE safety standard requires that the motor be deactivated when the safety guard is opened. By using the VRDY OFF Alarm Ignore signal, however, the CNC can be restarted without resetting, even if the safety guard has been opened.

Signal

All-axis VRDY OFF alarm ignore signal IGNVRY<Gn066.0>

[Classification] Input signal

[Function] Disables the detection of servo alarm (SV0401), VRDY OFF, for all axes.

[Operation] When this signal is set to logical 1, the control unit operates as follows:

- The control unit does not issue servo alarm (SV0401), VRDY OFF, even when the servo amplifier ready signal goes off. The control unit, however, sets servo ready signal SA to 0. The SA signal can remain set to 1, depending on the setting of parameter SAK (No. 1804#6).

Each-axis VRDY OFF alarm ignore signal IGVRY1 to IGVRY5 <Gn192.0 to Gn192.4>

[Classification] Input signal

[Function] Disables the detection of servo alarm (SV0401), VRDY OFF, for the corresponding axis. These signals correspond to the controlled axes. The suffixed number of each signal corresponds to the number of the controlled axis.

[Operation] When this signal is set to logical 1, the control unit operates as follows:

- The control unit does not issue servo alarm (SV0401), VRDY OFF, even when the servo amplifier ready signal for the corresponding axis goes off. The servo ready signal SA, however, is set to 0. The SA signal, however, can remain set to 1 depending on the setting of parameter SAK (No. 1804#6).

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn066								IGNVRY
Gn192				IGVRY5	IGVRY4	IGVRY3	IGVRY2	IGVRY1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1804		SAK		IVO				

[Input type] Parameter input

[Data type] Bit path

#4 IVO When an attempt is made to release an emergency stop while the VRDY OFF alarm ignore signal is 1:

0: The emergency stop state is not released until the VRDY OFF alarm ignore signal is set to 0.

1: The emergency stop state is released.

NOTE

When a reset is issued while the VRDY OFF alarm ignore signal is set to 1 and the motor activating current is low, the reset state can also be released, provided this parameter is set to 1.

#6 SAK When the VRDY OFF alarm ignore signal IGNVRY is 1, or when the each-axis VRDY OFF alarm ignore signals IGVRYn are 1:

0: Servo ready signal SA is set to 0.

1: Servo ready signal SA remains set to 1.

Alarm and message

Number	Message	Description
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.

Caution

⚠ CAUTION

1 When the control enters NOT READY status due to emergency stop or a servo alarm and then the control is reset, reset processing is not terminated until the VRDY OFF alarm ignore signal is set to 0.

2 When the VRDY OFF alarm ignore signal is set to 1 and the servo amplifier ready signal is set to off, the motor is freed from the drive, but follow up is not performed. To perform follow up, set the servo off signal to 1.

Note

NOTE

While the VRDY OFF alarm ignore signal is set to 1, and a servo alarm other than alarm PS0401 occurs, the control unit detects the alarm.

2.9 UNEXPECTED DISTURBANCE TORQUE DETECTION FUNCTION

Machine collision, defective, and damaged cutters cause a large load torque on the servo and spindle motors, compared with normal rapid traverse or cutting feed. This function detects the disturbance torque on the motors and sends this value as an estimated load torque to the PMC. If the detected disturbance torque value is abnormally great compared with the value specified in the parameter, the function stops the servo/spindle motor as early as possible or reverses the motor by an appropriate value specified in a parameter, in order to minimize possible damage to the machine. (The function to reverse motors is effective only for servo motors.)
The unexpected disturbance torque detection function is further divided as follows:

(1) Estimated disturbance torque output function
The CNC is always calculating the estimated disturbance torque for the motor (excluding acceleration/deceleration torque). The estimated disturbance torque output function enables the PMC to read the calculated torque using the window function.
(2) Unexpected disturbance torque detection alarm function
This function stops motors or reverses them by an amount specified in a parameter, causing the CNC to output an alarm, whenever the disturbance torque is greater than the value specified in a parameter. (The function to reverse motors is effective only for servo motors.)
(3) Unexpected disturbance torque detection group function
This function allows you to divide servo axes into desired groups. When the disturbance torque obtained by the estimated disturbance torque output function is greater than a parameter-set value, this function stops motors immediately. The function immediately stops all axes that are set for the same group by parameter setting (including axes of which group number set to 0) then places them in the interlock state. When a value is set in parameter No. 2103, this function reverses the motors by the set amount in a direction opposite to the advance direction, then places all the axes of the same group in the interlock state.

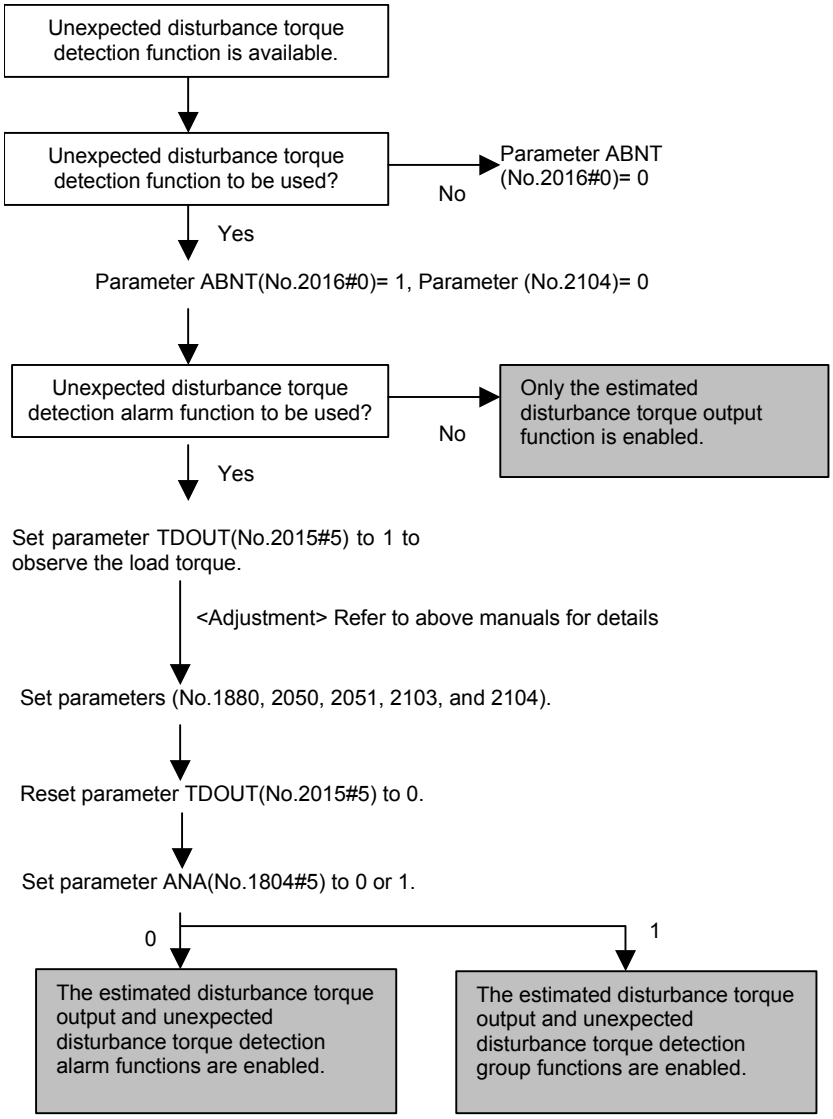
NOTE
Select the unexpected disturbance torque detection alarm function or unexpected disturbance torque detection group function by setting bit 5 (ANA) of parameter No. 1804.
You cannot use these functions at the same time.

Unexpected disturbance torque detection can also be disabled only for specific axes by using bit 5 (ABDSW) of parameter No. 2215 for the unexpected disturbance torque detection function and unexpected disturbance torque detection ignore signals IUDD1 to IUDD5 <G0125>. (This function is effective only for servo motors.)

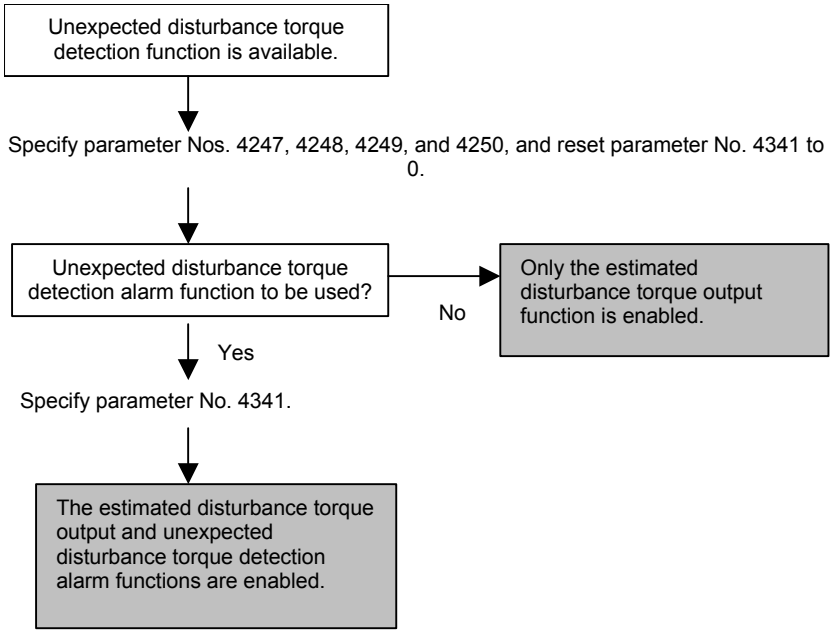
Procedure for parameter setting

The following flowcharts explain how to specify parameters for the unexpected disturbance torque detection function.
Refer to following manuals for details :
“FANUC AC SERVO MOTOR αi series PARAMETER MANUAL (B-65270EN)”
“FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)”

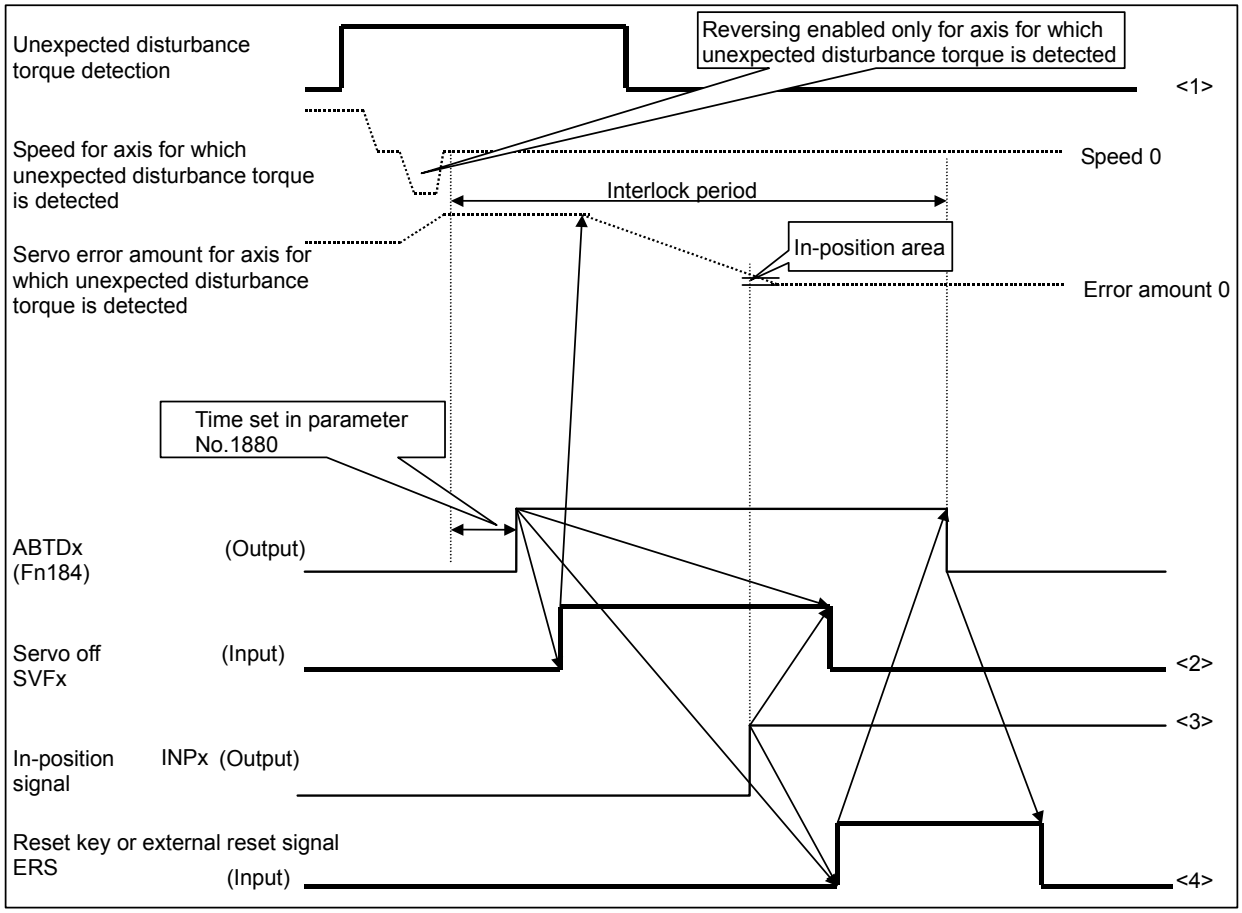
(1) Servo axis



(2) Spindle



・Timing chart when the unexpected disturbance torque detection group function is used



Timing chart of the unexpected disturbance torque detection group function

<1> When an unexpected disturbance torque is detected, servo motors are stopped immediately. Axes with the same group number set in parameter No. 1881 and axes with 0 set in the parameter are all stopped immediately then placed in the interlock state. At this time, axis movements are stopped regardless of commands of the CNC, so there is a difference between the coordinates held by the CNC and the actual motor position. This difference is accumulated as a servo error amount.

<2> When the operation is restarted, a follow-up operation is performed by inputting the servo off signal to match the coordinates held by the CNC and the actual motor position. Input the servo off signals for all axes of the same group (the axes with the same group number set and axes with group number 0 set). When servo off signals are input, the follow up signal *FLWU <G007.5> must be set to 0, and bit 0 (FUPx) of parameter No. 1819 must be set to 0.

<3> After servo off signal input, in-position signal INPx is turned on when the error amount enters the effective area.

<4> Check that the in-position signals for all axes that belong to the same group have been turned on, then input a reset (ERS or a reset key).

⚠ WARNING

The unexpected disturbance torque detection group function uses the servo off state. In the servo off state, the motor is deactivated and the dynamic brake does not work. This leaves the servo motor completely free with no braking power. Therefore, the vertical axis may drop freely in the event of a failure of the mechanical break or its drive circuit, sequence, etc. To detect an unexpected disturbance torque of the vertical axis, use the unexpected disturbance torque detection alarm function.

⚠ CAUTION

When an unexpected disturbance torque is detected using the unexpected disturbance torque detection group function, be sure to follow the timing chart shown above to input servo off signals then perform a reset operation. If a reset is made without inputting the servo off signals, a shock may apply to the machine.

NOTE

1 The servo off signal must be input with follow up signal *FLWU <G007.5> set to 0 and bit 0 (FUPx) of parameter No. 1819 set to 0.
2 Input the servo off signals for all axes of the group to which the axis with an unexpected disturbance torque detected belongs. Also for axes for which 0 is set in parameter No. 1881, input the corresponding servo off signals.
3 When an unexpected disturbance torque is detected for a PMC axis or when an axis for which an unexpected disturbance torque is detected belongs to a group that includes a PMC axis, all axes of the cluster to which that PMC axis belongs are placed in the interlock state.
4 When an unexpected disturbance torque is detected for a spindle (serial spindle), ABTSP1 or ABTSP2 <F0090.1 or .2> is output immediately. The setting of parameter No. 1880 is ignored.
5 When an unexpected disturbance torque is detected for a spindle (serial spindle), servo axes are not stopped. When the servo axes must be stopped, stop them by using ladder.

Signal

Servo axis unexpected disturbance torque detection signal ABTQSV <Fn090.0>

[Classification] Output signal
[Function] Informs the PMC that an unexpected disturbance torque was detected on a servo axis.
[Output cond.] This signal becomes "1" if:
- An Unexpected disturbance torque is detected for a servo axis, Cs axis, spindle positioning axis, or spindle axis during rigid tapping.

First-spindle unexpected disturbance torque detection signal ABTSP1 <Fn090.1>

[Classification] Output signal
[Function] Informs the PMC that an unexpected disturbance torque was detected on the first spindle axis.
[Output cond.] This signal becomes "1" if:
- An unexpected disturbance torque is detected for the first spindle under speed control.

Second-spindle unexpected disturbance torque detection signal ABTSP2 <Fn090.2>

[Classification] Output signal

[Function] Informs the PMC that an unexpected disturbance torque was detected on the second spindle axis.

[Output cond.] This signal becomes "1" if:

- An unexpected disturbance torque is detected for the second spindle under speed control.

Unexpected disturbance torque detection signals ABDT1 to ABDT5 <Fn184.0 to Fn184.4>

[Classification] Output signal

[Function] Notifies the PMC of the axis on which an unexpected disturbance torque was detected. The number at the end indicates the control axis number.

[Output cond.] This signal becomes "1" if:

- An unexpected disturbance torque is detected. (At this time, the servo axis unexpected disturbance torque detection signal ABTQSV<Fn090.0> also becomes "1".)

The following list summarizes the alarms and signals output by each function.

	Signal output		Alarm	
	ABTQSV	ABTSP1 ABTSP2	SV0409	SP0754
Servo axis	O	-	O	-
Cs contour control	O	-	O	-
Spindle positioning axis	O	-	O	-
Rigid tapping	O	-	-	O
Spindle axis for speed control	-	O	-	O

Unexpected disturbance torque detection disable signal IUDD1 to IUDD5 <Gn125.0 to Gn125.4>

[Classification] Input signal

[Function] Disables the unexpected disturbance torque detection function for each axis. This signal is provided for each control axis and the value at the end indicates the control axis number.

[Output cond.] If this signal is 1, the unexpected disturbance torque of the target axis is not detected.
Before using this signal, set bit 5 (ABDSW) of the parameter No. 2215, used for the unexpected disturbance torque detection function, to 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn090						ABTSP2	ABTSP1	ABTQSV
Fn184				ABDT5	ABDT4	ABDT3	ABDT2	ABDT1

	#7	#6	#5	#4	#3	#2	#1	#0
Gn125				IUDD5	IUDD4	IUDD3	IUDD2	IUDD1

Parameters

- Common to servo axis and spindle

	#7	#6	#5	#4	#3	#2	#1	#0
1804			ANA					

[Input type] Parameter input
[Data type] Bit path

#5 ANA When an unexpected disturbance torque is detected for an axis:
0: Movement along all axes is stopped, and a servo alarm is output.
1: No servo alarm is output, and movement along only the axes of the group containing the axis with the unexpected disturbance torque is stopped in interlock mode. Each unexpected disturbance torque detection signal is output. The unexpected disturbance torque detection signal that has been output is cleared by a reset. The group number of each axis is set in parameter No.1881.

> ⚠ **WARNING**
> The abnormal load detection group function uses the servo-off state in which the motor is de-energized and the dynamic brake does not operate. Accordingly, the servo motor enters the free running state and no braking force is applied. Therefore, for a vertical axis, if a failure occurs in the mechanical brake, driving circuit, or sequence, the axis may fall freely in a significant manner. When applying abnormal load detection to a vertical axis, use the abnormal load detection function.

> **NOTE**
> When this parameter is set to 1, it is necessary to set parameter No. 1881 to the group number of each axis when abnormal load is detected.

	#7	#6	#5	#4	#3	#2	#1	#0
1805				TSM	TSA			

[Input type] Parameter input
[Data type] Bit path

#3 TSA As the unexpected disturbance torque detection level during dwell, M code execution, and automatic operation halt state:
0: The threshold value for rapid traverse is used. (Parameter No. 2142)
1: The threshold value for cutting feed is used. (Parameter No. 2104)
When the parameter ABG0 (No. 2200#3) is se to 1, this parameter is enabled.

#4 TSM As the unexpected disturbance torque detection level in the jog feed mode (excluding manual rapid traverse) and manual handle feed mode:
0: The threshold value for rapid traverse is used. (Parameter No. 2142)
1: The threshold value for cutting feed is used. (Parameter No. 2104)
When the parameter ABG0 (No. 2200#3) is se to 1, this parameter is enabled.

1880	Unexpected disturbance torque detection alarm timer

[Input type] Parameter input
[Data type] Word path

[Unit of data] msec
[Valid data range] 0 to 32767
This parameter sets the time from the detection of an unexpected disturbance torque until a servo alarm is issued.
When 0 is set, however, the specification of 200 msec is assumed.

1881	Group number when an unexpected disturbance torque is detected

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 32
This parameter sets the group number of each axis, used when an unexpected disturbance torque is detected.
If an unexpected disturbance torque is detected for an axis, only the movement along the axes of the group containing the axis with the unexpected disturbance torque is stopped. If 0 is set for an axis, movement along that axis is stopped whenever an unexpected disturbance torque is detected for any axis.
This parameter is enabled when the parameter ANA (bit 5 of No.1804) is 1.
[Example]
When the settings indicated below are made, and an abnormal load is detected on the 3rd axis, the movements on the 1st axis, 2nd axis, 3rd axis, and 4th axis are stopped. When an abnormal load is detected on the 4th axis, the movements on the 2nd axis and the 4th axis are stopped.

Parameter No. 1881	Setting value
(1st axis)	1
(2nd axis)	0
(3rd axis)	1
(4th axis)	0
(5th axis)	2

- Servo axis

	#7	#6	#5	#4	#3	#2	#1	#0
2015			TDOUT					

[Input type] Parameter input
[Data type] Bit axis

#5 TDOUT Select output to the check board (for each axis)
0: Output the torque command to the check board.
1: Output the estimated load torque to the check board.

	#7	#6	#5	#4	#3	#2	#1	#0
2016								ABNT

[Input type] Parameter input
[Data type] Bit axis

#0 ABNT Unexpected disturbance torque detection function (for each axis) is
0: Disabled
1: Enabled
When using the unexpected disturbance torque detection alarm function and estimated disturbance torque output function, enable the unexpected disturbance torque detection function.

2050 **Velocity control observer**

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 32767
Be sure to set this parameter to 3559.
When using the velocity loop observer (by setting bit 2 (OBEN) of parameter No. 2003 to 1), set 956 in this parameter.

2051 **Velocity control observer**

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 32767
Be sure to set this parameter to 3329.
When using the velocity loop observer (by setting bit 2 (OBEN) of parameter No. 2003 to 1), set 510 in this parameter.

2103 **Retraction distance upon the detection of an unexpected disturbance torque**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] 0 to 32767
This parameter specifies the distance by which the tool is to be retracted, by reversing the motor, if an unexpected disturbance torque is detected.
When the motor is rotating at low speed, however, the tool may be retracted too far. To prevent this, the motor is stopped, instead of being reversed, while the specified feedrate is less than the value listed in the table below.
When this parameter is set to value A, the detection of an unexpected disturbance torque causes the tool to be retracted in the reverse direction by the distance A, then stopped, if the specified feedrate is equal to or greater than the value listed below.

Detection unit	Feedrate
10 μm	A / 0.8 mm/ min
1 μm	A / 8 mm/ min
0.1 μm	A / 80 mm/ min

When this parameter is set to 0, the motor stops immediately upon the detection of unexpected disturbance torque.

2104 **Threshold for unexpected disturbance torque detection alarm**

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Torque command unit
[Valid data range] 0 to 7282 (The maximum motor torque is 7282, regardless of the motor type.)
This parameter specifies the threshold load torque at which an Unexpected disturbance torque detection alarm is issued.
Monitor the load torque by setting parameter TDOUT (No.2015#5) to 1 then, for this parameter (No. 2104), set a value larger than the maximum monitored torque. An output of 4.4 V is equivalent to 7282 in the units of this parameter.

2142 **Threshold for unexpected disturbance torque detection alarm for rapid traverse**

[Input type] Parameter input

[Data type] Word axis
[Unit of data] Torque command unit
[Valid data range] 0 to 7282 (The maximum motor torque is 7282, regardless of the motor type.)
This parameter specifies the threshold load torque at which an Unexpected disturbance torque detection alarm is issued.
Monitor the load torque by setting parameter TDOUT (No.2015#5) to 1 then, for this parameter (No. 2104), set a value larger than the maximum monitored torque. An output of 4.4 V is equivalent to 7282 in the units of this parameter.
This parameter is enabled when ABG0 (bit 3 of parameter No. 2200) = 1.

	#7	#6	#5	#4	#3	#2	#1	#0
2200					ABG0			

[Input type] Parameter input
[Data type] Bit axis

#3 ABG0 The unexpected disturbance torque detection function for cutting or rapid traverse is:
0: Disabled.
1: Enabled.
When this function is disabled, the threshold value during cutting feed is used (parameter No. 2104).

	#7	#6	#5	#4	#3	#2	#1	#0
2215			ABDSW					

[Input type] Parameter input
[Data type] Bit axis

#5 ABDSW The Unexpected disturbance torque detection function for a specified axis:
0: Cannot be disabled.
1: Can be disabled.

- Parameters for spindle

	#7	#6	#5	#4	#3	#2	#1	#0
4015							SPLDMT	

[Input type] Parameter input
[Data type] Bit spindle

#1 SPLDMT Spindle load torque monitor function
0: The spindle load torque monitor function is disabled.
1: The spindle load torque monitor function is enabled.

4247	Magnetic flux compensation time constant for spindle load torque monitor

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] msec
[Valid data range] 0 to 8192
This parameter is used to compensate the delay in the generation of magnetic flux in the spindle motor relative to the specified value. When 0 is set, it is assumed that the generation of magnetic flux is not delayed.
Standard setting depends of the motor model.

4248	Spindle load torque monitor constant

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 32767
This constant is determined by the maximum output torque and inertia of the motor. It is used for observer processing.
Standard setting depends of the motor model.

4249	Observer gain 1 for spindle load torque monitor

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 32767
Standard setting is 500.

4250	Observer gain 2 for spindle load torque monitor

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 32767
Standard setting is 500.

4341	Threshold for unexpected disturbance torque detection alarm

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 0.01%
[Valid data range] 0 to 10000
This parameter specifies the threshold load torque at which an Unexpected disturbance torque detection alarm is issued for the spindle. Set a percentage (in units of 0.01 %) for the maximum output torque of the motor. When 0 is set, no Unexpected disturbance torque detection alarm is issued for the spindle.

Alarm and message

- Alarm related to servo axis

Number	Message	Description
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.

- Alarm related to spindle

Number	Message	Description
SP0754	ABNORMAL TORQUE	An abnormal load was detected in a spindle motor. The alarm can be canceled by RESET.

2.10 MACHINING CONDITION SELECTION FUNCTION

Overview

For high precision and high speed functions (advanced preview control, AI advanced preview control, AI contour control and AI contour control II functions), parameters which place importance on feedrate or precision are set and a precision level is specified according to the machining condition during machining. By using these parameters and level, parameters which satisfy the condition can automatically be calculated for machining.

NOTE
Machining condition selection function is optional function.

Format

- Changing the precision level by programming

The precision level can be changed not only by using the precision level selection screen but also by programming the following format:

M

- Selecting the level when the AI advanced preview control, AI contour control or AI contour control II mode is specified

G05.1 Q1 Rx ;
x : Level (1 to 10)

T

- Selecting the level when the advanced preview control mode is specified

G08 P1 Rx ;
x : Level (1 to 10)

⚠ CAUTION
Once a level is specified, it is kept valid even after the AI contour control/AI contour control II mode is canceled.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
13600								MCR
	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 (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 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.

NOTE
The acceleration rate change time ranges from 0 to 127 msec when this parameter is 0 or ranges from 0 to 200 msec when this parameter is 1.

	#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 when this parameter is set to 1, the precision level selection screen is displayed.

13610	Acceleration rate for acceleration/deceleration before look-ahead interpolation in advanced preview control/AI advanced preview control/AI contour control/AI contour control II (precision level 1)
13611	Acceleration rate for acceleration/deceleration before look-ahead interpolation in advanced preview control/AI advanced preview control/AI contour control/AI contour control II (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 advanced preview control/AI advanced preview control/AI contour control/AI contour control II. 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/AI contour control II is used (precision level 1)
13613	Acceleration rate change time (bell-shaped) when AI contour control/AI contour control II 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/AI contour control II. Set a value (precision level 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

NOTE
These parameters are valid when bit 7 (MSA) of parameter No. 13600 is 0.

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 II.
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 II.
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 II 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 II 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 II.
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 advanced preview control/AI advanced preview control/AI contour control/AI contour control II is used (precision level 1)**
13621	**Allowable acceleration rate when advanced preview control/AI advanced preview control/AI contour control/AI contour control II 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 advanced preview control/AI advanced preview control/AI contour control/AI contour control II. 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 advanced preview control/AI advanced preview control/AI contour control/AI contour control II is used (precision level 1)
13623	Time constant for acceleration/deceleration after interpolation when advanced preview control/AI advanced preview control/AI contour control/AI contour control II 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 advanced preview control/AI advanced preview control/AI contour control/AI contour control II 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 advanced preview control/AI advanced preview control/AI contour control/AI contour control II is used (precision level 1)
13625	Corner speed difference when advanced preview control/AI advanced preview control/AI contour control/AI contour control II 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 advanced preview control/AI advanced preview control/AI contour control/AI contour control II. 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 advanced preview control/AI advanced preview control/AI contour control/AI contour control II is used (precision level 1)
13627	Maximum cutting speed when advanced preview control/AI advanced preview control/AI contour control/AI contour control II 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 advanced preview control/AI advanced preview control/AI contour control/AI contour control II. 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 advanced preview control/AI advanced preview control/AI contour control/AI contour control II is used
13629	Parameter number corresponding to arbitrary item 2 when advanced preview control/AI advanced preview control/AI contour control/AI contour control II 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 (No. 4000 to No. 4799)
- Parameters of real number type
- Parameters that require power-off (for which the alarm (PW0000) is issued)
- Nonexistent parameters

13630	**Value with emphasis on speed (precision level 1) of the parameter corresponding to arbitrary item 1 when advanced preview control/AI advanced preview control/AI contour control/AI contour control II is used**
13631	**Value with emphasis on speed (precision level 1) of the parameter corresponding to arbitrary item 2 when advanced preview control/AI advanced preview control/AI contour control/AI contour control II is used**
13632	**Value with emphasis on speed (precision level 10) of the parameter corresponding to arbitrary item 1 when advanced preview control/AI advanced preview control/AI contour control/AI contour control II is used**
13633	**Value with emphasis on speed (precision level 10) of the parameter corresponding to arbitrary item 2 when advanced preview control/AI advanced preview control/AI contour control/AI contour control II 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.

13634	**Precision level currently selected when advanced preview control/AI advanced preview control/AI contour control/AI contour control II is used**

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to 10
This parameter sets the level currently selected.

13662	
	Acceleration rate change time (bell-shaped) when AI contour control/AI contour control II is used (precision level 1), range extended

13663	Acceleration rate change time (bell-shaped) when AI contour control/AI contour control II 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/AI contour control II. Set a value (precision 1) with emphasis placed on speed, and a value (precision level 10) with emphasis on precision.

NOTE
These parameters are valid when bit 7 (MSA) of parameter No. 13600 is 1.

2.11 MACHINING QUALITY LEVEL ADJUSTMENT (M Series)

M

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 optional.

Format

- Changing the precision level by a program

This specification is the same as that of the conventional machining condition selection function.
For information on the precision level, see Section 2.10, "MACHINING CONDITION SELECTION FUNCTION".

- 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 Nano smoothing mode is canceled.

Standard level

When the parameters Nos. 11686 and 11687 are set to the default smoothing level and precision level, respectively, a return to the default levels in the parameters Nos. 11686 and 11687 is made during power-off or a reset.
Set the default levels to an integer from 1 to 10.

When the default level is set to 0, the current level is maintained.

Limitation

- This function requires the following options.
  - AI contour control II
  - Nano smoothing
  - Machining condition selection function
- In the following cases, the machining quality level adjustment screen is not displayed. Instead, the precision level selection screen and smoothing level selection screen are displayed as the machining level selection screen.
  - Bit 6 (QLS) of parameter No. 11350 is set to 0.
  - A LCD other than 10.4-inch is used.
- On the screen of the machining quality level adjustment function, the PMC single axis is not displayed and adjustment is impossible.

Notes

- The RMS value of each item changes in the following cases.
  - The precision level or smoothing level is changed.
  - The value of the level-1 parameter or level-10 parameter of each item of the precision level or smoothing level is changed. (A change by soft key [INIT] or [GROUP INIT] is included.)
- When the parameter corresponding to the RMS value is changed directly, the new value is used instead of the value obtained from the present level and the level-1 and level-10 setting values of the precision level and smoothing level.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
11318								
			LSO					

[Input type] Parameter input
[Data type] Bit

#5 **LSO** When level is selected on precision level select screen or machining quality level adjustment screen:
0: Soft key and MDI key can be used.
1: Only soft key can be used.

	#7	#6	#5	#4	#3	#2	#1	#0
11350								
		QLS						

[Input type] Parameter input
[Data type] Bit

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#6 QLS The machining quality level adjustment screen is:
0: Not displayed.
1: Displayed.

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 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 level 1 and level 10.

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 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 II 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 II 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.

2.12 MALFUNCTION PREVENT FUNCTIONS

Overview

These functions monitor the CNC internal status and check that related data is within the allowable range. If an invalid state due to a deteriorated hardware component or noise is detected, these functions stop the machine with an alarm to prevent any malfunction.

Explanation

Each function is detailed below.

- Checking the maximum speed of the servo motor

This function checks whether the maximum speed specified in parameter No. 12255 is exceeded. If the maximum speed is exceeded, this function issues alarms DS0004 and PW0006 and stops the machine. When the parameter is set to 0, this function checks whether the maximum speed (999000 for IS-B) is exceeded.

- Checking the maximum acceleration of the servo motor

This function checks whether the maximum acceleration specified in parameter No. 12256 is exceeded. If the maximum acceleration is exceeded, this function issues alarms DS0005 and PW0006 and stops the machine. When the parameter is set to 0, this function does not check the maximum acceleration.

- Checking the maximum speed of the spindle motor

This function checks whether the maximum speed of the spindle motor is exceeded. If the maximum speed is exceeded, this function issues alarms IE0007 and PW0006 and outputs S0. The parameter which specifies the maximum speed of each spindle is used.

- Checking the stored stroke limit at the end point

This function makes a stored stroke check immediately before pulses are output in addition to a normal stored stroke check. If the stored stroke limit is exceeded, this function issues one of alarms IE0001 to IE0006 and alarm PW0006 and stops the machine. The parameters for normal stored stroke checks are used.

- Monitoring execution of NC command analysis

This function monitors NC command analysis to check the execution counter and reference area for validity. If invalid data is detected, this function issues one of alarms DS0006 to DS0013 and alarm PW0006 and stops the machine.

- Monitoring execution of acceleration/deceleration after interpolation

This function monitors the execution counter during execution of linear/bell-shaped acceleration/deceleration after interpolation to check the counter for validity. If invalid data is detected, this function issues alarms IE0008 and PW0006 and stops the machine.

Parameter

12255	Maximum servo motor speed

[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 maximum servo motor speed. When the value set in this parameter is exceeded, the servo motor stops with the alarm (DS0004). When 0 is set in this parameter, the specification of a maximum allowable value (999000 for IS-B) is assumed.

12256	Maximum servo motor acceleration rate

[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, 0.0 to +10000.0)
This parameter sets a maximum servo motor acceleration rate. When the value set in this parameter is exceeded, the servo motor stops with the alarm (DS0005). When 0 is set in this parameter, alarm check is not performed.

Alarm and message

The machine is decelerated and stopped when a DS alarm is issued or is immediately stopped when an IE alarm is issued to prevent any malfunction.
Check the block in which the machine is stopped. If you cannot find the cause, contact FANUC without releasing the alarm state.

Number	Message	Description
DS0004	EXCESS MAXIMUM FEEDRATE	The malfunction prevention function detected the command in which a value exceeding the maximum speed was specified.
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 illegal reference area.
DS0011	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an illegal reference area.
DS0012	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an illegal reference area.
DS0013	ILLEGAL REFERENCE AREA	The malfunction prevention function detected an illegal reference area.
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.

2.13 OPERATOR ERROR PREVENT FUNCTIONS

Overview

If an invalid value is set for tool offset or the machine is operated improperly, a workpiece may be cut incorrectly or a tool may be damaged. If data is lost due to an operator error, time is required for recovery.

The operator error prevent functions include the following functions to prevent an unintentional operator operation (called an operator error below).

1. Functions operating during data setting
   - Checking whether offset data is within the valid setting range
   - Reconfirming incremental input operation
   - Disabling absolute input using soft keys to prevent an operator error related to absolute and incremental input
   - Reconfirming deletion of a program or all data
   - Reconfirming data update during setting
2. Functions operating during execution
   - Highlighting updated modal data
   - Displaying the status of the execution block before execution of a program
   - Displaying the axis status such as the mirror image enable state or interlock enable state
   - Checking the start of a program from a midpoint
   - Checking whether offset data is within the valid setting range
   - Checking the maximum incremental amount

Explanation

- Functions operating during data setting

The following functions are provided to prevent an operation error during data setting.

- Checking of the input data range
- Reconfirming of incremental input
- Prohibiting of soft key absolute input
- Reconfirming of program deletion
- Reconfirming of all data deletion
- Reconfirming of data update during setting

For details on these functions, refer to the Operator's Manual (B-64304EN).

- Functions related to checking before execution

The following functions are provided to prevent an operation error during execution.

- Update display of modal information
- Start check signal
- Status display of axes
- Reconfirming of midway block start
- Checking of the execution data range
- Checking of the maximum incremental amount

For details on the functions other than the start check signal, refer to the Operator's Manual (B-64304EN).

- Start check signal

If cycle start is performed when the start check signal STCHK <Gn408.0> is ON, the stop state is entered with the remaining traverse distance and modal of the execution block indicated. Performing cycle start again starts the execution.

An operation error is prevented because the status of the execution block can be checked before execution.
Using the start check signal together with the update display of modal information shown above makes checking easier.

Signals

Start check signal STCHK<Gn408.0>

[Classification] Input signal
[Function] The traverse distance and modal can be checked before execution.
[Operation] When this signal is 1, the remaining traverse distance and modal is displayed before executing a block and then execution stops. Performing cycle start again starts the execution.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn408								STCHK

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3109						IKY		

[Input type] Parameter input
[Data type] Bit path

#2 IKY On the tool offset screen and workpiece shift screen (T series), soft key [INPUT] is:
0: Displayed.
1: Not displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3191					SSF			
					SSF	WSI		

[Input type] Parameter input
[Data type] Bit path

#2 WSI On the workpiece zero point offset screen, the soft key [INPUT] is:
0: Displayed.
1: Not displayed.

#3 SSF On the setting screen, the soft key for confirming data input is:
0: Not displayed.
1: Displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
8136		NGW						

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#6 NGW Tool offset memory C (M series) or tool geometry/wear compensation (T series) is:
0: Used.
1: Not Used.

10000	**Lower limit 1 of tool offsets No.01**
to	
10019	**Lower limit 1 of tool offsets No.20**

[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 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 lower limits of the following offset values:

- T series, without tool geometry/wear offsets, X-axis offset
- T series, with tool geometry/wear offsets, X-axis and geometry offsets
- M series, tool offset memory A, offset
- M series, tool offset memory C, geometry and length offsets

10020	**Upper limit 1 of tool offsets No.01**
to	**to**
10039	**Upper limit 1 of tool offsets No.20**

[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 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 upper limits of the following offset values:

- T series, without tool geometry/wear offsets, X-axis offset
- T series, with tool geometry/wear offsets, X-axis and geometry offsets
- M series, tool offset memory A, offset
- M series, tool offset memory C, geometry and length offsets

10040	**Lower limit 2 of tool offsets No.01**
to	**to**
10059	**Lower limit 2 of tool offsets No.20**

[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 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 lower limits of the following offset values:

- T series, without tool geometry/wear offsets, Z-axis offset
- T series, with tool geometry/wear offsets, Z-axis and geometry offsets
- M series, tool offset memory C, geometry and radius offsets

10060	Upper limit 2 of tool offsets No.01
to	to
10079	Upper limit 2 of tool offsets No.20

[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 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 upper limits of the following offset values:

- T series, without tool geometry/wear offsets, Z-axis offset
- T series, with tool geometry/wear offsets, Z-axis and geometry offsets
- M series, tool offset memory C, geometry and radius offsets

10080	Lower limit 3 of tool offsets No.01
to	to
10099	Lower limit 3 of tool offsets No.20

[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 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 lower limits of the following offset values:

- T series, without tool geometry/wear offsets, tool nose radius offset
- T series, with tool geometry/wear offsets, tool nose radius and geometry offsets

10100	Upper limit 3 of tool offsets No.01
to	to
10119	Upper limit 3 of tool offsets No.20

[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 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 upper limits of the following offset values:

- T series, without tool geometry/wear offsets, tool nose radius offset
- T series, with tool geometry/wear offsets, tool nose radius and geometry offsets

10120	Lower limit 4 of tool offsets No.01
to	to
10139	Lower limit 4 of tool offsets No.20

[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 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 lower limits of the following offset values:

- T series, with tool geometry/wear offsets, X-axis and wear offsets
- M series, tool offset memory C, wear and length offsets

10140	Upper limit 4 of tool offsets No.01
to	to
10159	Upper limit 4 of tool offsets No.20

[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 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 upper limits of the following offset values:

- T series, with tool geometry/wear offsets, X-axis and wear offsets
- M series, tool offset memory C, wear and length offsets

10160	Lower limit 5 of tool offsets No.01
to	to
10179	Lower limit 5 of tool offsets No.20

[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 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 lower limits of the following offset values:

- T series, with tool geometry/wear offsets, Z-axis and wear offsets
- M series, tool offset memory C, wear, and radius offsets

10180	Upper limit 5 of tool offsets No.01
to	to
10199	Upper limit 5 of tool offsets No.20

[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 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 upper limits of the following offset values:

- T series, with tool geometry/wear offsets, Z-axis and wear offsets
- M series, tool offset memory C, wear, and radius offsets

10200	Lower limit 6 of tool offsets No.01
to	to
10219	Lower limit 6 of tool offsets No.20

[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 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 lower limits of the following offset values:

- T series, with tool geometry/wear offsets, tool nose radius and wear offsets

10220	**Upper limit 6 of tool offsets No.01**
to	**to**
10239	**Upper limit 6 of tool offsets No.20**

[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 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 upper limits of the following offset values:

- T series, with tool geometry/wear offsets, tool nose radius and wear offsets

10240	**Lower limit 1 of a tool offset number range No.01**
to	**to**
10259	**Lower limit 1 of a tool offset number range No.20**

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to maximum number of offset sets
Each of these parameters sets the lower limit of a tool offset number range.
These parameters correspond to the tool offset lower/upper limits set in parameter Nos. 10000 to 10239.

10260	**Upper limit 1 of a tool offset number range No.01**
to	**to**
10279	**Upper limit 1 of a tool offset number range No.20**

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to maximum number of offset sets
Each of these parameters sets the upper limit of a tool offset number range.
These parameters correspond to the tool offset lower/upper limits set in parameter Nos. 10000 to 10239.

10280	**Lower limit 7 of tool offsets No.01**
to	**to**
10283	**Lower limit 7 of tool offsets No.04**

[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 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 lower limits of the following offset values:

- T series, without tool geometry/wear offsets, Y-axis offset
- T series, with tool geometry/wear offsets, Y-axis and geometry offsets

10284	Upper limit 7 of tool offsets No.01
to	to
10287	Upper limit 7 of tool offsets No.04

[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 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 upper limits of the following offset values:

- T series, without tool geometry/wear offsets, Y-axis offset
- T series, with tool geometry/wear offsets, Y-axis and geometry offsets

10288	Lower limit 8 of tool offsets No.01
to	to
10291	Lower limit 8 of tool offsets No.04

[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 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 lower limits of the following offset values:

- T series, with tool geometry/wear offsets, Y-axis and wear offsets

10292	Upper limit 8 of tool offsets No.01
to	to
10295	Upper limit 8 of tool offsets No.04

[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 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 upper limits of the following offset values:

- T series, with tool geometry/wear offsets, Y-axis and wear offsets

10296	Lower limit 2 of a tool offset number range No.01
to	to
10299	Lower limit 2 of a tool offset number range No.04

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to maximum number of offset sets
Each of these parameters sets the lower limit of a tool offset number range.
These parameters correspond to the tool offset lower/upper limits set in parameter Nos. 10280 to 10295.

10300	Upper limit 2 of a tool offset number range No.01
to	to
10303	Upper limit 2 of a tool offset number range No.04

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to maximum number of offset sets
Each of these parameters sets the upper limit of a tool offset number range.
These parameters correspond to the tool offset lower/upper limits set in parameter Nos. 10280 to 10295.

10304	Lower limit of workpiece zero point offsets No.01
to	to
10309	Lower limit of workpiece zero point offsets No.06

[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)
Each of these parameters sets the lower limit of workpiece zero point offset values.

10310	Upper limit of workpiece zero point offsets No.01
to	to
10315	Upper limit of workpiece zero point offsets No.06

[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)
Each of these parameters sets the upper limit of workpiece zero point offset values.

10316	Lower limit of a workpiece zero point offset range No.01
to	to
10321	Lower limit of a workpiece zero point offset range No.06

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to maximum number of offset sets
Each of these parameters sets the lower limit of a workpiece zero point offset range. For an additional workpiece coordinate system, set a value after adding 1000.
These parameters correspond to the workpiece zero point offset lower/upper limits set in parameter Nos. 10304 to 10315.

10322	Upper limit of a workpiece zero point offset range No.01
to	to
10327	Upper limit of a workpiece zero point offset range No.06

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to maximum number of offset sets

Each of these parameters sets the upper limit of a workpiece zero point offset range. For an additional workpiece coordinate system, set a value after adding 1000.
These parameters correspond to the workpiece zero point offset lower/upper limits set in parameter Nos. 10304 to 10315.

10328	Lower limit of workpiece shifts

[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)
This parameter sets a workpiece shift lower limit.

10329	Upper limit of workpiece shifts

[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)
This parameter sets a workpiece shift upper limit.

	#7	#6	#5	#4	#3	#2	#1	#0
10330		ASD	EBC	MID	HSC	ADC	PDC	IIC

[Input type] Parameter input
[Data type] Bit

#0 IIC At the time of incremental input, a confirmation message is:
0: Displayed.
1: Not displayed.

#1 PDC At the time of program deletion, a confirmation message is:
0: Displayed.
1: Not displayed.

#2 ADC At the time of deletion of all data, a confirmation message is:
0: Displayed.
1: Not displayed.

#3 HSC When a cycle start is executed halfway in the program, a confirmation message is:
0: Displayed.
1: Not displayed.

#4 MID Updated modal information is:
0: Highlighted.
1: Not highlighted.

#5 EBC Program sum checking is:
0: Disabled.
1: Enabled.

#6 ASD Axis state display is:
0: Enabled.
1: Disabled.

10331	Lower limit of external workpiece zero point offsets

[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 lower limit of external workpiece zero point offsets.

10332	Upper limit of external workpiece zero point offsets

[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 upper limit of external workpiece zero point offsets.

- Additional information

Parameters for input data range checks are assigned as follows:

- Tool offset
  <1> No tool geometry/wear offset for a T series
  Nos. 10000 to 10119, Nos. 10240 to 10279
  <2> With tool geometry/wear offset for a lathe system
  Nos. 10000 to 10279
  <3> Tool offset memory A for an M series
  Nos. 10000 to 10039, Nos. 10240 to 10279
  <4> Tool offset memory C (geometry/wear, length/radius) for an M series
  Nos. 10000 to 10079, Nos. 10120 to 10199, Nos. 10240 to 10279
- Y-axis tool offset
  Nos. 10280 to 10303
- Workpiece origin offset
  Nos. 10304 to 10327
- Workpiece shift
  Nos. 10328 and 10329

Alarm and message

Number	Message	Description
PS0334	OFFSET IS OUT OF EFFECTIVE RANGE	An offset data which was out of the effective range was specified. (operator error prevention function)
PS0337	EXCESS MAXIMUM INCREMENTAL VALUE	The command value exceeded the maximum amount of incremental. (operator error prevention function)

3 MANUAL OPERATION

Chapter 3, “MANUAL OPERATION”, consists of the following sections:

3.1 JOG FEED/INCREMENTAL FEED....256
3.2 MANUAL HANDLE FEED....262
3.3 MANUAL HANDLE INTERRUPT....271
3.4 I/O Link β MANUAL HANDLE INTERFACE....276
3.5 MANUAL HANDLE RETRACE....279

3.1 JOG FEED/INCREMENTAL FEED

Overview

- Jog feed

In jog mode, setting a feed axis and direction selection bit to "1" on the machine operator's panel moves the tool along the selected axis in the selected direction.
Manual operation is allowed one axis at a time. Up to 3 axes can be selected at a time by setting parameter JAX (No.1002#0).

- Incremental feed

In incremental feed mode, setting a feed axis and direction selection bit to "1" 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. The step can be 10, 100, or 1000 times the least input increment.
Bit 2 (HNT) of parameter No. 7103 can be used to multiply the obtained step by 10.
The jog feedrate is specified in parameter (No.1423)
The jog feedrate can be adjusted with the jog feedrate override dial.
With rapid traverse selected, the tool can be moved at the rapid traverse rate regardless of the jog feedrate override signal.

Signal

The following signals determine that way the jog feed or incremental feed is executed.

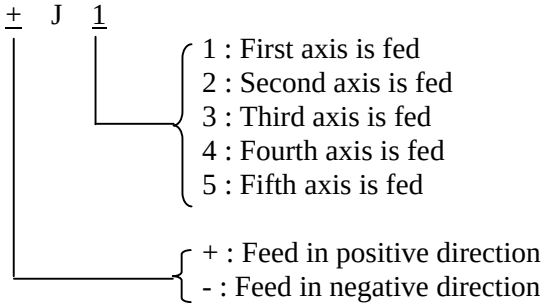
Selection	Jog feed	Incremental feed
Mode selection	MD1, MD2, MD4, MJ	MD1, MD2, MD4, MINC
Selection of the axis to move	+J1, -J1, +J2, -J2, +J3, -J3, ...	
Selection of the direction to move the axis		
Selection of the move amount		MP1, MP2
Selection of feedrate	*JV0 to *JV15, RT, ROV1, ROV2	

The only difference between jog feed and incremental feed is the method of selecting the feed distance. In jog feed, the tool continues to be fed while the following signals selecting the feed axis and direction are "1": +J1, -J1, +J2, -J2, +J3, -J3, etc. In incremental feed, the tool is fed by one step.
The distance of the step is selected by the manual handle feed move distance selection signals MP1 and MP2.
For the signals selecting the mode, see "Mode Selection Signals." For the manual handle feed selection signals, MP1 and MP2 select the move amount, "Manual handle feed." For rapid traverse override signals ROV1 and ROV2, see “Feedrate Override Signals.”
Other signals are described below.

Feed Axis and Direction Selection Signal +J1 to +J5<Gn100.0 to Gn100.4>, -J1 to -J5<Gn102.0 to TGn102.4>

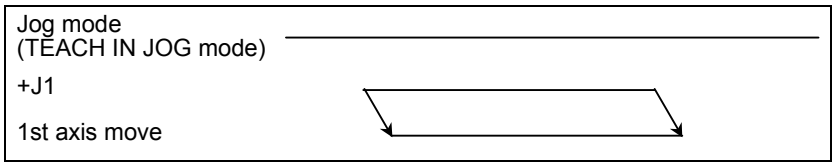
[Classification] Input signal

[Function] Selects a desired feed axis and direction in jog feed or incremental feed. The sign (+ or -) in the signal name indicates the feed direction. The number following J indicates the number of the control axis.



[Operation] When the jog bit is "1", the control unit operates as described below.

- When jog feed or incremental feed is allowed, the control unit moves the specified axis in the specified direction. In jog feed, the control unit continues to feed the axis while the bit is "1".

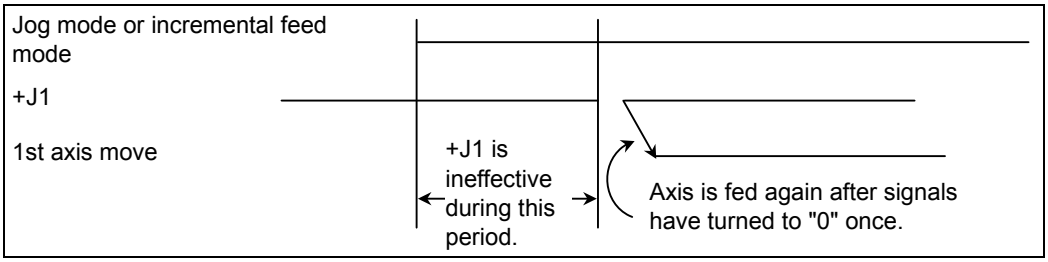


- In incremental feed, the control unit feeds the requested axis by the step distance which is specified by the manual handle feed move distance selection signals MP1, MP2, then the axis stops. Even if the signal is set to "0" while the axis is being fed, the control unit does not stop moving.
  To feed the axis again, set the signal to "0", then to "1" again.

⚠ CAUTION

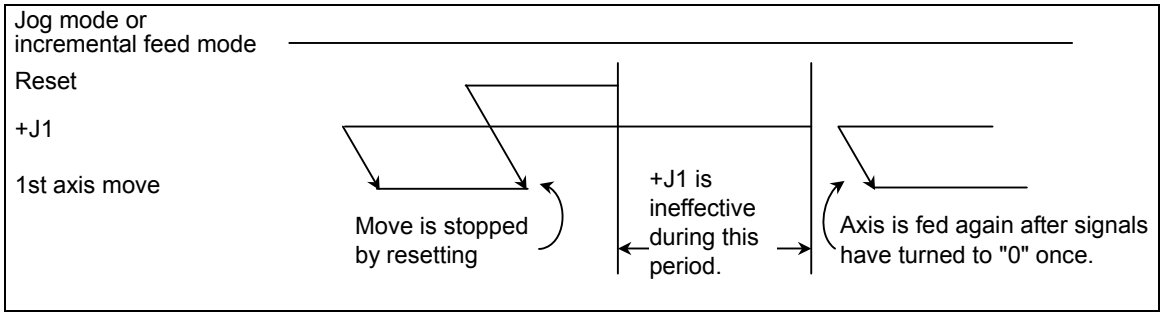
1 If both the positive direction and negative direction signals of the same axis are simultaneously set to "1", neither the positive direction nor the negative direction is selected. The control unit assumes that both these signals are set to "0".

2 If the feed axis and direction selection signals are set to "1" before the jog feed mode or incremental feed mode is selected, these signals are invalidated. After the jog feed mode or incremental feed mode is selected, set these signal to "0", then set them to "1" again.



⚠ CAUTION

3 If the control unit is reset while the feed axis and direction selection signals are set to "1" or if a feed axis and direction signal turns to "1" while the control unit is in the reset state, the signal will be ignored even after releasing reset. After the reset state is released, set these signals to "0", then set them to "1" again.



Manual feedrate override signal *JV0 to *JV15<Gn010,Gn011>

[Classification] Input signal

[Function] Selects a feedrate in jog feed or incremental feed. These signals are in sixteen bit binary code, which corresponds to the override values as follows:

$$\text{Override value}\,(\%) = 0.01\% \times \sum_{i=0}^{15} \left| 2^i \times Vi \right|$$

where Vi=0 when *JVi is 1.
Vi=1 when *JVi is 0.

The override value is assumed to be zero when all of the signals, (*JV0 to *JV15) are set to "1" or "0". When this occurs, the feed is stopped. The override value can be specified in the range of 0% to 655.34% in units of 0.01%. Some examples are listed below.

*JV0 to *JV15				Override value
12	8	4	0	
1 1 1 1	1 1 1 1	1 1 1 1	1 1 1 1	0
1 1 1 1	1 1 1 1	1 1 1 1	1 1 1 0	0.01
1 1 1 1	1 1 1 1	1 1 1 1	0 1 0 1	0.10
1 1 1 1	1 1 1 1	1 0 0 1	1 0 1 1	1.00
1 1 1 1	1 1 0 0	0 0 0 1	0 1 1 1	10.00
1 1 0 1	1 0 0 0	1 1 1 0	1 1 1 1	100.00
0 1 1 0	0 0 1 1	1 0 1 1	1 1 1 1	400.00
0 0 0 0	0 0 0 0	0 0 0 0	0 0 0 1	655.34
0 0 0 0	0 0 0 0	0 0 0 0	0 0 0 0	0

[Operation] If rapid traverse selection signal RT is "0" during jog feed or incremental feed, the manual feedrate specified by parameter (No. 1423) is overridden by the value specified by the *JVi signals.

NOTE

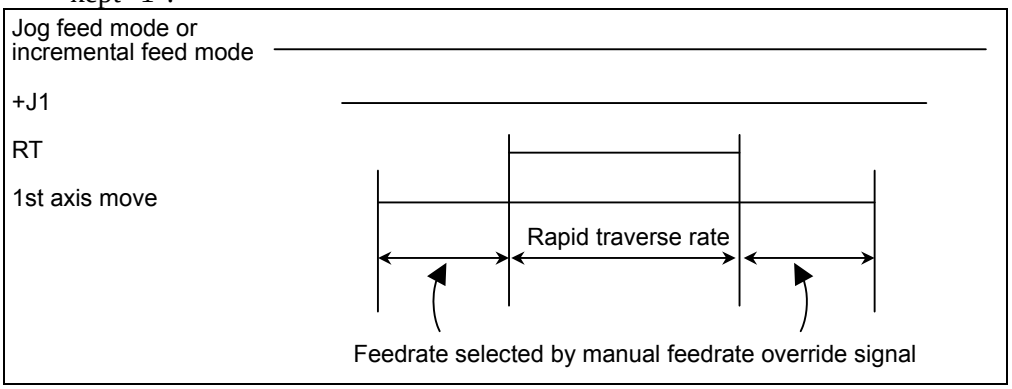
The *JVi signals also serve as the override signals during dry run in automatic operation mode.

Manual rapid traverse selection signal RT<Gn019.7>

[Classification] Input signal

[Function] Selects a rapid traverse rate for jog feed or incremental feed.

[Operation] When the signal turns to "1", the control unit operates as described below:
The control unit executes the jog feed or incremental feed at a rapid traverse rate. The rapid traverse override is validated.
When the signal is switched from "1" to "0" or vice versa during jog feed or incremental feed, the feedrate is decelerated until it reaches zero, then increased to the specified value. During acceleration and deceleration, the feed axis and direction selection signal can be kept "1".



⚠ WARNING
After the power is turned on, the stroke limit function does not work until the reference position return is completed. During this period, the control unit ignores the RT signal, if it is set to "1", and keeps moving the tool at a feedrate selected by the manual feedrate override signal. The parameter RPD (No. 1401#0) can be set so the rapid traverse is validated before the reference position return is completed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn010	*JV7	*JV6	*JV5	*JV4	*JV3	*JV2	*JV1	*JV0
Gn011	*JV15	*JV14	*JV13	*JV12	*JV11	*JV10	*JV9	*JV8
Gn019	RT							
Gn100				+J5	+J4	+J3	+J2	+J1
Gn102				-J5	-J4	-J3	-J2	-J1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002								JAX

[Input type] Parameter input
[Data type] Bit path

#0 JAX Number of axes controlled simultaneously in manual continuous feed, manual rapid traverse and manual reference position return
0: 1 axis
1: 3 axes

	#7	#6	#5	#4	#3	#2	#1	#0
1401								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

	#7	#6	#5	#4	#3	#2	#1	#0
1402				JRV			JOV	

[Input type] Parameter input
[Data type] Bit path

#1 JOV Jog override is:
0: Enabled
1: Disabled (tied to 100%)

#4 JRV Jog feed or incremental feed is
0: Performed at feed per minute.
1: Performed at feed per rotation.

NOTE
Specify a feedrate in parameter No.1423.

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 JRV, bit 4 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 JRV, bit 4 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 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.

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx				

[Input type] Parameter input
[Data type] Bit axis

#4 JGLx Acceleration/deceleration in jog feed
0: Exponential acceleration/deceleration is applied.
1: Depending on which is used for cutting feed. (depending on the setting in CTBx and CTLx, bit 1 and 0 of parameter No.1610)

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.

1625	FL rate of exponential acceleration/deceleration in 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)
Set the FL rate of exponential acceleration/deceleration in cutting feed for each axis. This parameter allows only the exponential type.

	#7	#6	#5	#4	#3	#2	#1	#0
7103						HNT		

[Input type] Parameter input
[Data type] Bit path

#2 HNT When compared with the travel distance magnification selected by the manual handle feed travel distance selection signals (incremental feed signals) (MP1, MP2), the travel distance magnification for incremental feed/manual handle feed is:
0: Same.
1: 10 times greater.

Warning

⚠ WARNING
For incremental feeding along an axis under diameter programming, the tool moves in units of the diameter.

Note

NOTE
1 Time constant and method of automatic acceleration/deceleration for manual rapid traverse are the same as G00 in programmed command.
2 If a manual pulse generator is provided, the manual handle feed mode is enabled instead of incremental feed mode. However, using parameter JHD (bit 0 of parameter No. 7100) enables both manual handle and incremental feed in the manual handle feed mode.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Jog feed
	Incremental feed

3.2 MANUAL HANDLE FEED

Overview

In the handle mode, when the manual pulse generator on the machine operator's panel is rotated, the axis moves accordingly. 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.
The distance the tool is to be move can be multiplied by one of 4 magnifications selected by MP1, MP2<Gn019.4,5>. The distance can be further multiplied by 10 using bit 2 (HNT) of parameter No. 7103.
The number of manual pulse generators is as follows.
(M series)
Max. 3
(T series)
Max. 2 (Max. 3 is an option function for 0*i*-TD)

NOTE
To use manual handle feed, set 1 in bit 0 (HPG) of parameter No. 8131.

Explanation

- **Manual handle feed in the jog feed mode**
- **Incremental feed in the manual handle feed mode**

The following states can be selected using bit 0 (JHD) of parameter No. 7100.

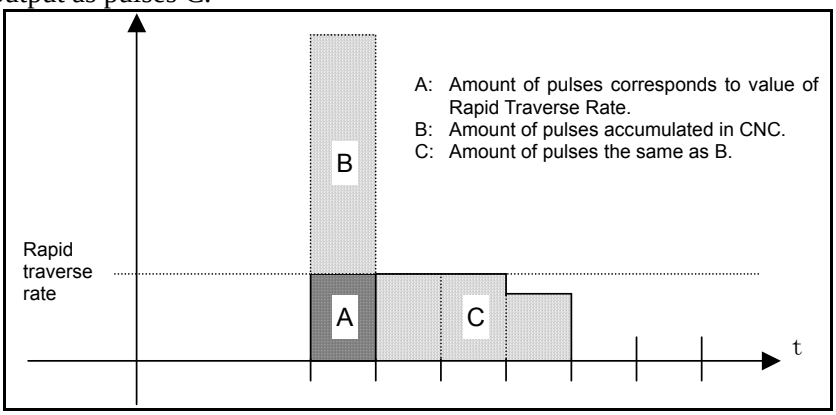
	JHD=0		JHD=1	
	Jog feed mode	Manual handle feed mode	Jog feed mode	Manual handle feed mode
Jog feed	Enabled	Disabled	Enabled	Disabled
Manual handle feed	Disabled	Enabled	Enabled	Enabled
Incremental feed	Disabled	Disabled	Disabled	Enabled

- Manual handle feed in TEACH IN JOG mode

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.



Amount of pulses output by CNC in Manual Handle Feed

Amount of pulses B is calculated in 2 cases as following:

In case of

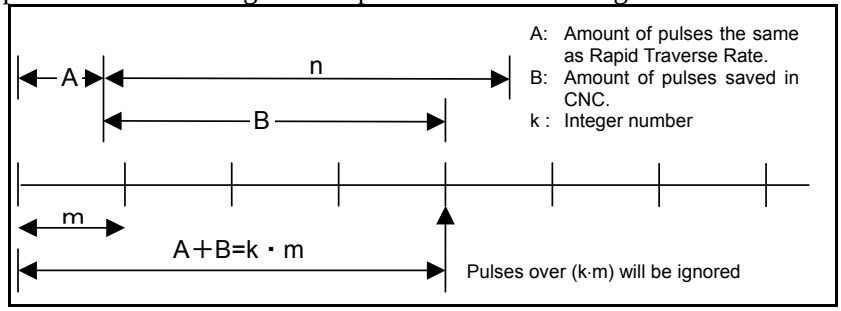
1) Parameter No.7117 = 0
   The feedrate is clamped at the Rapid Traverse Rate and generated pulses exceeding the Rapid Traverse Rate are ignored (B=0).

In case of

2) 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 MP1, MP2<Gn019.4,5> 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, shown 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.



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 limit of the feedrate in manual handle feed

The upper limit of the feedrate is determined as follows depending on the input signal (maximum manual handle feedrate switch signal HNDLF) from PMC.

- If HNDLF is 0, clamping is performed at the manual rapid traverse rate (parameter No. 1424).
- If HNDLF is 1, clamping is performed at the rate set in parameter No. 1434.

- Movement direction of an axis to the rotation of MPG

Parameter HNGx (No. 7102 #0) 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.

Manual handle feed direction inversion signal HDN<Gn347.1> inverts the axis movement direction in relation to the rotation direction of the manual handle. The axis whose movement direction is to be inverted can be selected using bit 1 (HNAx) of parameter No. 7102.

The parameter and signal take effect only on the following functions:

- Manual handle feed
- Manual handle interruption

- Per-axis magnification setting

By setting arbitrary values in parameter Nos. 12350 and 12351 for manual handle feed maginifications m and n, you can set a specific maginification for each axis.
When no value is set in parameter No. 12350, parameter No. 7113 is used. When no value is set in parameter No. 12351, parameter No. 7114 is used.
These parameters take effect for manual handle interrupts as well.

Warning

⚠ 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

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.

Signal

Manual Handle Feed Axis Selection Signals

HS1A to HS1D<Gn018.0 to 3>
HS2A to HS2D<Gn018.4 to 7>
HS3A to HS3D<Gn019.0 to 3>

[Classification] Input signal

[Function] Selects the axis of manual handle feed. A set of four code signals, A, B, C, and D is provided for each manual pulse generator. (Up to 3 generators can be used.) (For multi-path control, there is a set of code signals in each path.) The number in the signal name indicates the number of the manual pulse generator to be used.

HSxA

x : 1 1st. manual pulse generator
2 2nd. manual pulse generator
3 3rd. manual pulse generator

Code signals A, B, C, D, and E correspond to the feed axes listed in following table:

Manual handle feed axis selection signals				Feed axis
HSnD	HSnC	HSnB	HSnA	
0	0	0	0	No selection (None of axis is fed)
0	0	0	1	1st axis
0	0	1	0	2nd axis
0	0	1	1	3rd axis
0	1	0	0	4th axis
0	1	0	1	5th axis

Manual handle feed amount selection signals (Incremental feed signals)

MP1,MP2<Gn019.4,5>
MP21,MP22<Gn087.0,1>
MP31,MP32<Gn087.3,4>

[Classification] Input signal

[Function] This signal selects the distance traveled per pulse from the manual pulse generator during the manual handle feed or manual handle interrupt. It also selects the distance traveled per incremental feed step.

The table below lists the signal-to-distance correspondence.

Manual Handle Feed Amount Selection signals		Distance Traveled		
MP2	MP1	Manual Handle Feed	Manual Handle Interruption	Incremental Feed
0	0	Minimum unit × 1	Minimum unit × 1	Minimum unit × 1
0	1	Minimum unit × 10	Minimum unit × 10	Minimum unit × 10
1	0	Minimum unit × m*1	Minimum unit × m*1	Minimum unit × 100
1	1	Minimum unit × n*1	Minimum unit × n*1	Minimum unit × 1000

*1 Scale factors m and n are specified using parameter No.7113 and 7114.
Scale factors m and n of each axis are specified using parameter No.12350 and 12351.

⚠ CAUTION
1 Because the least input increment is used as the units for manual handle and incremental feed, the same value represents a different distance depending on whether the metric or inch input system is used.
2 For an axis under diameter programming, the tool moves by the diameter value.

Furthermore, Manual Handle Feed Amount Selection Signals of each Manual Pulse Generator can be used by setting bit 5 (MPX) of parameter No.7100. The relation between Manual Handle Feed Amount Selection signals of each Manual Pulse Generator and parameter to set magnification is showed in the next table.

Value of Parameter MPX (No.7100#5)	Manual Pulse Generator	Manual Handle Feed Amount Selection signals	Parameter of Magnification	
			m_x	n_x
MPX=0	1st.-3rd. Manual Pulse Generator	MP1,MP2	No.7113	No.7114
MPX=1	1st. Manual Pulse Generator	MP1,MP2	No.7113	No.7114
	2nd. Manual Pulse Generator	MP21,MP22	No.7131	No.7132
	3rd. Manual Pulse Generator	MP31,MP32	No.7133	No.7134

Maximum manual handle feedrate switch signal HNDLF<Gn023.3>

[Classification] Input signal

[Function] Selects the maximum feedrate in manual handle feed. The maximum feedrate in manual handle feed is determined according to the state of this signal as follows.

Maximum manual handle feedrate switch signal	Maximum feedrate
0	Manual rapid traverse rate (parameter No. 1424)
1	Maximum manual handle feedrate (parameter No. 1434)

Manual handle feed direction inversion signal HDN<Gn347.1>

[Classification] Input signal

[Function] Inverts the rotation direction of the manual pulse generator and the axis movement direction in manual handle feed.

0: Does not invert the axis movement direction in relation to the rotation direction of the manual pulse generator.

1: Inverts the axis movement direction in relation to the rotation direction of the manual pulse generator.

The axis whose movement direction is to be inverted by this signal can be selected using bit 1 (HNAx) of parameter No. 7102. This signal does not work on the rotation direction of manual linear/circular interpolation.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn018	HS2D	HS2C	HS2B	HS2A	HS1D	HS1C	HS1B	HS1A
Gn019			MP2	MP1	HS3D	HS3C	HS3B	HS3A
Gn023					HNDLF			
Gn087				MP32	MP31		MP22	MP21
Gn347							HDN	

Parameter

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.

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. This value is effective when the maximum manual handle feedrate switch signal HNDLF is 1.

	#7	#6	#5	#4	#3	#2	#1	#0
7100			MPX				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
0: Invalid
1: Valid

	When JHD:=0		When JHD:=1	
	JOG feed mode	Manual handle feed mode	JOG feed mode	Manual handle feed mode
JOG feed	○	×	○	×
Manual handle feed	×	○	○	○
Incremental feed	×	×	×	○

#1 THD In the TEACH IN JOG mode, the manual pulse generator is:
0: Disabled.
1: Enabled.

#5 MPX In Manual handle feed mode, manual handle feed amount selection signal is
0 : same for all manual pulse generator, and it is set by signals MP1 and MP2<Gn019.4,5>.
1 : differ to each other manual pulse generator, and it's setting signal as follow:
1st. Manual Pulse Generator : MP1,MP2<Gn019.4,5>
2nd. Manual Pulse Generator : MP21,MP22<Gn087.0,1>
3rd. Manual Pulse Generator : MP31,MP32<Gn087.3,4>

	#7	#6	#5	#4	#3	#2	#1	#0
7102							HNAx	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

#1 HNAx When manual handle feed direction inversion signal HDN<Gn347.1> is set to "1", the direction of movement is set for each axis with respect to the rotation direction of the manual pulse generator.
0: The axis movement direction is the same as the direction in which the manual pulse generator rotates.
1: The axis movement direction is opposite to the direction in which the manual pulse generator rotates.
When the rotation direction is reversed by manual handle feed direction inversion signal HDN<Gn347.1>, the rotation axis direction obtained by the setting of bit 0 (HNGx) of parameter No. 7102 is reversed.

	#7	#6	#5	#4	#3	#2	#1	#0
7103					HNT			

[Input type] Parameter input
[Data type] Bit path

#3 HNT When compared with the travel distance magnification selected by the manual handle feed travel distance selection signals (incremental feed signals) (MP1, MP2), the travel distance magnification for manual handle interrupt is:
0: Same.
1: 10 times greater.

	#7	#6	#5	#4	#3	#2	#1	#0
7105							HDX	

[Input type] Parameter input
[Data type] Bit

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#1 HDX Manual handle for I/O Link connection is:
0: Automatically set.
1: Manually set.

NOTE
In manual setting, parameters No.12300 to No.12302 must be set by manual to connect Manual Pulse Generator with I/O Link.

7113	Manual handle feed magnification m

[Input type] Parameter input

[Data type] Word path
[Valid data range] 1 to 2000
This parameter sets the magnification m when manual handle feed movement selection signals MP1 and MP2 are set to 0 and 1.

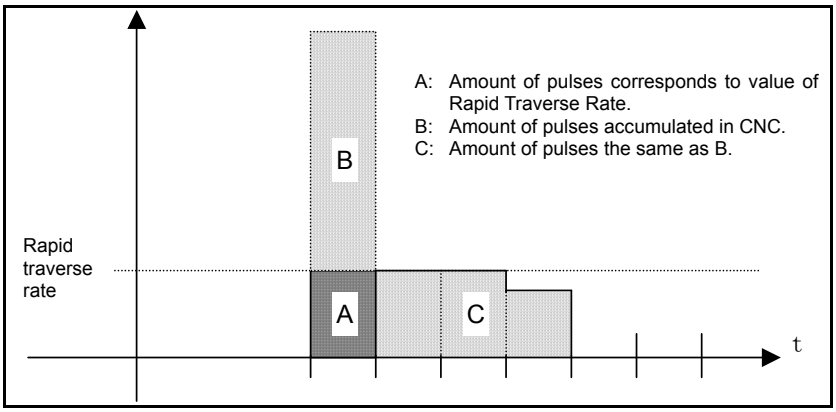
7114	Manual handle feed magnification n

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 2000
This parameter sets the magnification when manual handle feed movement selection signals MP1 and MP2 are set to 1.

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.

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.



Amount of pulses output by CNC in Manual Handle Feed

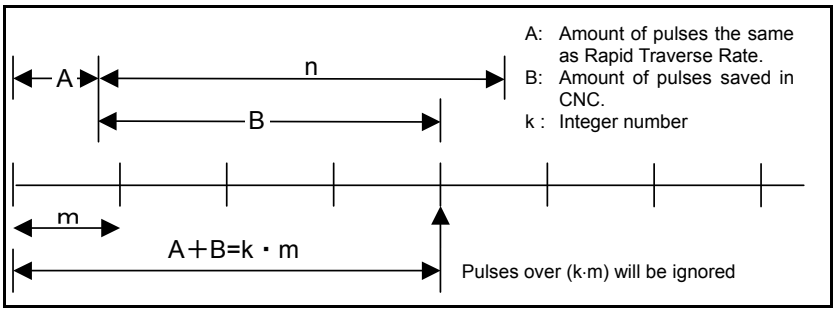
Amount of pulses B is calculated in 2 cases as following:

In case of
1) 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).

In case of
2) 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 MP1, MP2<Gn019.4,5> 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, shown 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.



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.

7131	**Manual handle feed magnification m2 / 2nd. manual pulse generator**
7132	**Manual handle feed magnification n2 / 2nd. manual pulse generator**
7133	**Manual handle feed magnification m3 / 3rd. manual pulse generator**
7134	**Manual handle feed magnification n3 / 3rd. manual pulse generator**

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 2000
The 'mx' is selected when MPx1 = 0, MPx2 = 1.
The 'nx' is selected when MPx1 =1, MPx2 = 1.

12300	**X address of the 1st. manual pulse generator**
12301	**X address of the 2nd. manual pulse generator**
12302	**X address of the 3rd. manual pulse generator**

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] -1, 0 to 127, 200 to 327
To set X address of manual pulse generator connected with I/O Link in PMC.
When the manual pulse generator is not connected, set -1 to this parameter.

	PMC Path	X Address
1st. manual pulse generator	No.12340	No.12300
2nd. manual pulse generator	No.12341	No.12301
3rd. manual pulse generator	No.12342	No.12302

NOTE
Set these parameters when bit 1 (HDX) of parameter No. 7105 is set to 1. When HDX = 0, these parameters are automatically set. If a manual handle is not connected when HDX = 0, -1 is set automatically.

12350	Manual handle feed magnification m in each axis

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 2000
For each axis, this parameter sets the magnification m when manual handle feed movement selection signals MP1 = 0, MP2 = 1.

NOTE
When value is set to 0 for this parameter, the parameter No. 7113 is valid.

12351	Manual handle feed magnification n in each axis

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 2000
For each axis, this parameter sets the magnification when manual handle feed movement selection signals MP1 = 1, MP2 = 1.

NOTE
When value is set to 0 for this parameter, the parameter No. 7114 is valid.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Manual handle feed

3.3 MANUAL HANDLE INTERRUPT

Overview

In the automatic operation mode (manual data input, DNC operation, or memory operation) or memory edit mode, you can turn the handle of the manual pulse generator to overlap the movement by manual handle operation with that by automatic operation. The axis to which the handle interrupt is applied is selected using the manual handle interrupt axis selection signal. The minimum travel distance per graduation is the least command increment. The distance can be scaled by one of four types of magnification that is selected using MP1 and MP2 <G019 .4 and .5>. The scaled distance can be multiplied by 10 using bit 3 (HIT) of parameter No. 7103. Each scale factor can be selected using the manual handle travel distance selection signals (refer to "Manual Handle Feed").

> ⚠ **WARNING**
> The amount of movement per graduation for manual handle interrupt is equal to the least input increment like that for manual handle feed. For example, for metric input on an inch machine, the tool moves by 0.01 inches for 254 graduations; for inch input on a millimeter machine, it moves by 0.254 mm for 100 graduations.

Signal

Manual Handle Interrupt Axis Selection Signals
HS1IA to HS1ID<Gn041.0 to 3>
HS2IA to HS2ID<Gn041.4 to 7>
HS3IA to HS3ID<Gn042.0 to 3>

[Classification] Input signal

[Function] These signals select an axis to which the manual handle interrupt is applied. There are four sets of signals, each corresponding to a manual pulse generator (up to three). Each set consists of five code signals A, B, C, and D. The number in the signal name indicates the number of the manual pulse generator to be used.

HSxIA

x : 1 Selects the axis for which manual pulse generator No. 1 is used
2 Selects the axis for which manual pulse generator No. 2 is used
3 Selects the axis for which manual pulse generator No. 3 is used

The correspondence between code signals A, B, C, and D and selected feed axes is similar to the correspondence of the manual handle feed axis selection signals. See "Manual Handle Feed Axis Selection signals."

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn041	HS2ID	HS2IC	HS2IB	HS2IA	HS1ID	HS1IC	HS1IB	HS1IA
Gn042					HS3ID	HS3IC	HS3IB	HS3IA

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1606								MNJx

[Input type] Parameter input
[Data type] Bit axis

#0 MNJx In manual handle interrupt:

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
6004						VHD		

[Input type] Parameter input
[Data type] Bit path

#2 VHD With system variables #5121 to #5125:
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 (bit 6 (NGW) of parameter No. 8136 is 0)).
1: An interrupt travel distance based on manual handle interrupt is read.

	#7	#6	#5	#4	#3	#2	#1	#0
7100					HCL			

[Input type] Parameter input
[Data type] Bit path

#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		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.

#3 HIT When compared with the travel distance magnification selected by the manual handle feed travel distance selection signals (incremental feed signals (MP1, MP2), the travel distance magnification for manual handle interrupt is:
0: Same.
1: 10 times greater.

7113	Manual handle feed magnification m

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 2000
This parameter sets the magnification m when manual handle feed movement selection signals MP1 and MP2 are set to 0 and 1.

7114	Manual handle feed magnification n

[Input type] Parameter input
[Data type] Word path
[Valid data range] 1 to 2000

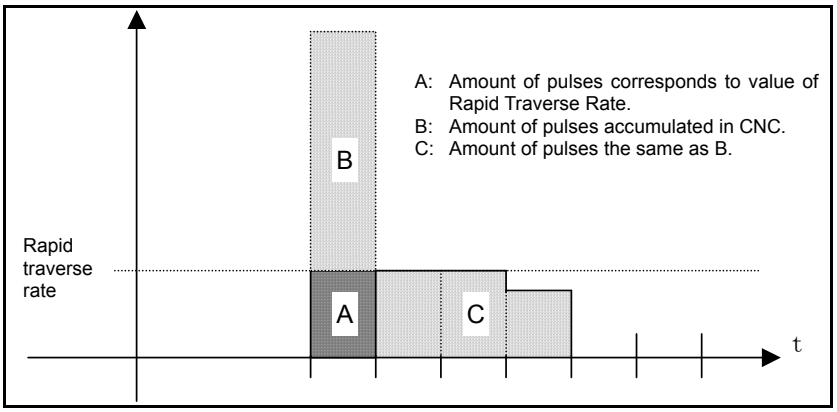
This parameter sets the magnification when manual handle feed movement selection signals MP1 and MP2 are set to 1.

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.

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.



Amount of pulses output by CNC in Manual Handle Feed

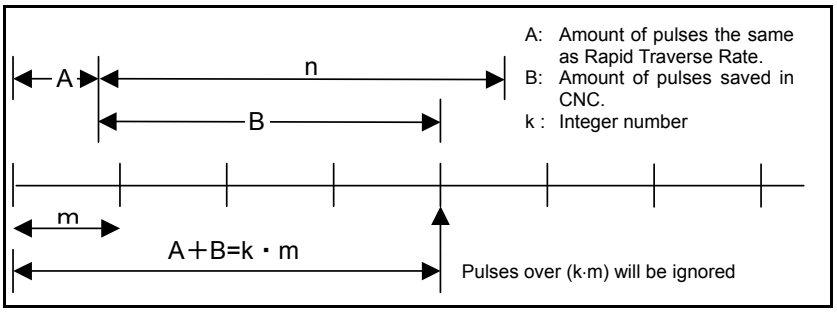
Amount of pulses B is calculated in 2 cases as following:

In case of
1) 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).

In case of
2) 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 MP1, MP2<Gn019.4,5> 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, shown 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.



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.

12350	Manual handle feed magnification m in each axis

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 2000
For each axis, this parameter sets the magnification m when manual handle feed movement selection signals MP1 and MP2 are set to 0 and 1.

NOTE
When no value is set for this parameter, use parameter No. 7113.

12351	Manual handle feed magnification n in each axis

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 2000
For each axis, this parameter sets the magnification when manual handle feed movement selection signals MP1 and MP2 are set to 1.

NOTE
When no value is set for this parameter, use parameter No. 7114.

Note

NOTE
1 In the manual operation mode (such as the jog feed mode, manual handle feed mode, or TEACH IN HANDLE mode), handle interrupt is disabled.
2 When the machine lock or interlock signal is on, the tool is not moved by handle interrupt.
3 Manual handle interrupt is not effective for an axis in either of the following states:
- Follow-up
- PMC axis control

4 Manual handle interrupt is not allowed for an axis specified in G00 mode.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Manual handle interrupt
CONNECTION MANUAL (FUNCTION) (B-64303EN-1) (This manual)	Manual handle feed

3.4 I/O Link β MANUAL HANDLE INTERFACE

Overview

This function controls manual handle feed for FANUC servo unit β*i* series with I/O Link option (I/O Link β*i*) by using a manual pulse generator on the host (CNC). Pulses from manual pulse generator are transferred from the host (CNC) to I/O Link β*i* through I/O Link. Still further, this function can control the magnification of pulses by changing the parameter. This function is available on the peripheral control interface.

Explanation

After the manual handle mode of I/O Link β*i* (MD1(Y0.0)=0), MD2(Y0.1)=0,MD4(Y0.2)=1) is selected, the host informs the magnification of the manual handle generator (MP1(Y7.4), MP2(Y7.5)), and changes the manual pulse counter.

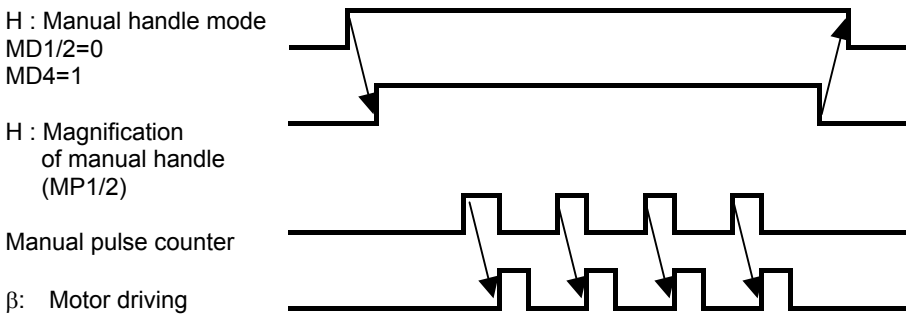
After reading the varied amount of the manual pulse counter, I/O Link β*i* operates the motor.

Actual pulses are that calculated pulses multiplied by the parameter (No.10809-10811).

【Timing chart】

H: The management of the host

β: The management of the I/O Link β*i*



> ⚠ **CAUTION**
> Please be sure to transfer 16 bytes signals from the host (CNC) to I/O Link β*i*.

Signal

Mode selection signal MD1 <Yy+0.0>, MD2 <Yy+0.1>, MD4<Yy+0.2>

[Classification] CNC(host) → I/O Link β*i*

[Function] This signal selects the operation mode of I/O Link β*i*.

[Operation] This signal selects the manual handle mode of the I/O Link β*i*.

MD4	MD2	MD1	Mode
1	0	0	Handle feed

> ⚠ **CAUTION**
> 1 Please use the manual pulse generator on the manual handle mode.
> 2 Please don't change the operation mode during the manual handle operation of the I/O Link β*i*.

Incremental feed signal MP1 <Yy+7.4>, MP2 <Yy+7.5>

[Classification] CNC(host) → I/O Link β*i*

[Function] This signal selects the distance traveled per a pulse from the manual pulse generator during the manual handle feed on the I/O Link β*i*.

[Operation] During the manual operation mode, the I/O Link β*i* operates the motor by reading the varied amount of the manual pulse counter and multiplying them by the value determined by this signal.

MP2	MP1	Distance traveled per a pulse from the manual pulse generator
0	0	User-specified unit × 1
0	1	User-specified unit × 10
1	0	User-specified unit × 100
1	1	User-specified unit × (M/N) (M : Parameter No.0062, N : Parameter No.0063)

> ⚠ **CAUTION**
> 1 These signals are available when the Parameter MP (No.0005#5) on the I/O Link β*i* is set to 1.
> 2 These signals are available on the manual handle mode only.
> 3 These signals are shared with the signals for rapid traverse override.
> 4 On manual handle mode, these signals mean Incremental Feed signals. On another mode, these signals mean Rapid Traverse Override signals. When you change the operation mode from the manual handle mode to another, you need to change these signals to the setting of Rapid Traverse Override.

Manual handle generators selection signal IOLBH1 <G199.0>, IOLBH2 <G199.1>

[Classification] Input signal (common among paths)

[Function] This signal selects the manual handle generator which operates the I/O Link β*i*.

[Operation] This signal selects the manual handle generator which operates the I/O Link β*i*.

IOLBH2	IOLBH1	Manual handle generator which operates I/O Link β*i*
0	0	First unit
0	1	Second unit
1	0	Third unit
1	1	Not available

> ⚠ **CAUTION**
> Please don't change the manual handle generator on the manual handle mode.

Signals on I/O Link *β* side

No.	#7	#6	#5	#4	#3	#2	#1	#0
Yy+0						MD4	MD2	MD1
Yy+7			MP2	MP1				

Signals on CNC

No.	#7	#6	#5	#4	#3	#2	#1	#0
G199							IOLBH2	IOLBH1

Parameter

- Parameter on I/O Link β*i* side

No.	#7	#6	#5	#4	#3	#2	#1	#0
005			MP	IOH				

[Data type] Bit

#4 IOH Manual handle feed by way of I/O Link is :
0 : Unavailable
1 : Available

> ⚠ **CAUTION**
> Please be sure to set the parameter EXPLS (No.0003#6) on the I/O Link β*i* to 0, when IOH is set to 1.

#5 MP On the manual handle feed, 4 kinds of magnifications set by the signal MP1/MP2 are :
0 : Unavailable
1 : Available

- Parameter on CNC side

> ⚠ **CAUTION**
> The Change of the parameter LBH (No.7105#5) and the parameter (No.12330 - 12337) is valid after you turn off the power and turn on again. For example : Even if you change the parameter LBH (No.7105#5) from 1(available) to 0(unavailable), the motor moves according the movement of the manual handle, till you turn off the power.

	#7	#6	#5	#4	#3	#2	#1	#0
7105			LBH					

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

#5 LBH Manual handle feed for the I/O Link β*i* using the I/O link manual pulse generator is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
12330	G17	G16	G15	G14	G13	G12	G11	G10
	#7	#6	#5	#4	#3	#2	#1	#0
12331	G1F	G1E	G1D	G1C	G1B	G1A	G19	G18
	#7	#6	#5	#4	#3	#2	#1	#0
12332	G27	G26	G25	G24	G23	G22	G21	G20
	#7	#6	#5	#4	#3	#2	#1	#0
12333	G2F	G2E	G2D	G2C	G2B	G2A	G29	G28

[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.

G10 to G2F When the Power Mate or I/O Link β*i* is connected to the I/O Link, these bits set whether to transfer pulses from manual pulse generators connected to the I/O Link to the Power Mate or I/O Link β*i*.

The setting of each bit has the following meaning:
0: Pulses are transferred.
1: Pulses are not transferred.

The bits and the corresponding I/O Link channel numbers and group numbers are listed below:

Parameter	Channel number	Group number
G10	1	0
G11	1	1
G12	1	2
:	:	:
G1F	1	15
:	:	:
G2F	2	15

NOTE
1 When one group is neither Power Mate CNC nor I/O Link β*i*, pulses from the manual pulse generator is not transferred to the group regardless of the parameter setting.
2 When the Power Mate is connected to the I/O Link, set 1 in this parameter.

3.5 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.

NOTE
Manual handle retrace function is optional functio.

- 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 the memory mode (MEM mode), and the checking mode signal MMOD<Gn067.2> 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 <Gn067.3> 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.

Explanation

- Control by the manual handle

Program execution start

The checking mode signal MMOD<Gn067.2> is set to "1" in the memory mode (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 <Gn067.3> 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<Gn067.3> is set to "0", it is controlled as usual execution.
When check mode signal MMOD<Gn067.2> 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<Fn091.3> 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 the parameter RPO(No.6400#0) 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 parameter RPO(No.6400#0) 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, 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.

- 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)

Milling system

```
G00  G01  G02  G03  G04  G22  G23
G28  G30  G40  G41  G42
G43  G44  G49  G53  G65  G73  G76
G80  G81  G82  G83  G85  G86  G87
G88  G89  G92  G94  G95  G96  G97
```

NOTE
1 In Small-Holl Pecking Drilling Cycle(G83), backward movement is prohibited.
2 In forward movement of Boring Cycle(G88), the sequence of actions at bottom of holl 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 parameter RVN(6400#5) 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:
No.6400#2=1, #3=0 (5 M-codes/group and 16 groups)
No.6411=100 ⎫
No.6412=101 ⎪
No.6413=102 ⎬ Group A
No.6414=103 ⎪
No.6415=104 ⎭
No.6416=200 ⎫
No.6417=201 ⎪
No.6418=202 ⎬ Group B
No.6419=203 ⎪
No.6420=204 ⎭

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 show as next table.

	Forward movement	Backward movement
O10;		
N1G4X1.;		
N2M101;	M101	M100 (*1)
N3G4X1.;		
N4M204;	M204	M200 (*1)
N5G4X1.;		
N6M104;	M104	M101 (*2)
N7G4X1.;		
N8M300;	M300	M300 (*3)

	Forward movement	Backward movement
N9G4X1.;		
N10M200;	M200	M204 (*2)
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 the parameter STO(No.6401#7) 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)

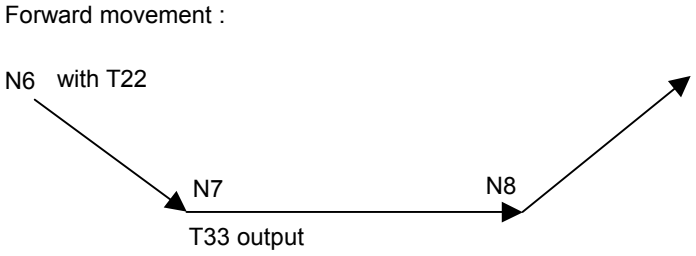
T-code output timing at the backward movement

T-code is output as follows when the program proceeds backward after the forward movement to N8 block.

	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.



Backward movement (When parameter STO is set to “0”) :



Backward movement (When parameter STO is set to “1”) :



- 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<Fn091.1>.

It becomes the change prohibition state under the following condition.

- While the axis is moved
- 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 the wait for block change (for T series two-path systems only; See "Waiting in a 2-path system" later in this section.)

- 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<Fn091.2>.
If the following blocks are executed in backward movement, backward movement is prohibited.

- Program number block of main program (except subprogram and macro program)
- 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.3.5(a). When the following conditions are not full, the clock is displayed.

- When parameter CHS(No.6401#2) is set to "0":
  1) Software option of handle manual retrace is enabled.
  2) Status display disable/enable parameter HST(No.6401#6) is set to "1".
  3) Check mode confirmation signal MMMOD<Fn091.3> is set to "1".

- When parameter CHS(No.6401#2) is set to "1":
  1) Software option of handle manual retrace is enabled.
  2) Status display disable/enable parameter HST(No.6401#6) is set to "1".
  3) Cycle start signal STL<Fn000.5> is set to "1".
  4) Check mode signal MMOD<Gn067.2> is set to "1".
  5) Manual handle check signal MCHK<Gn067.3> is set to "1".

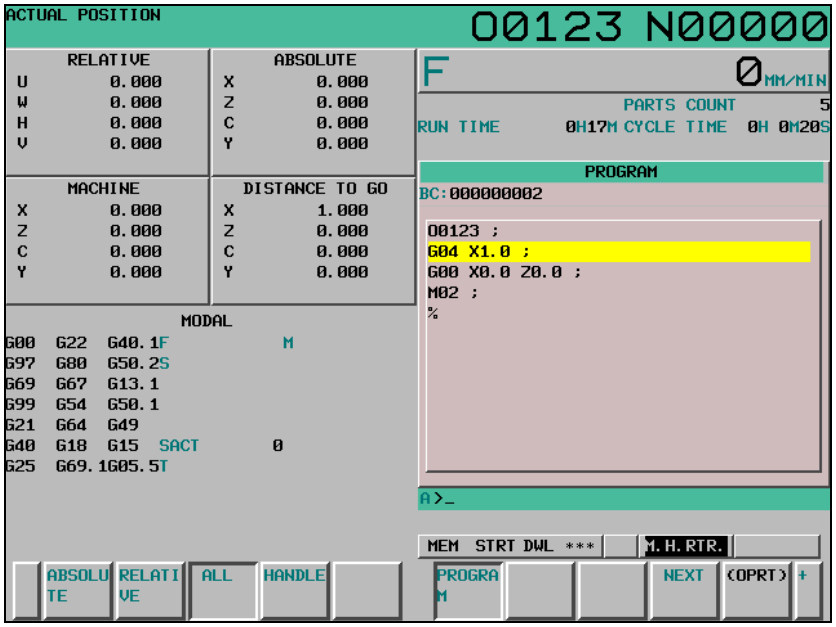


Fig. 3.5 (a) "M.H.RTR." status display

Besides, when reverse movement prohibition signal MRVSP<Fn091.2> 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.3.5(b). When reverse movement prohibition signal MRVSP<Fn091.2> is set to "0", the " M.H.RTR " is displayed again.

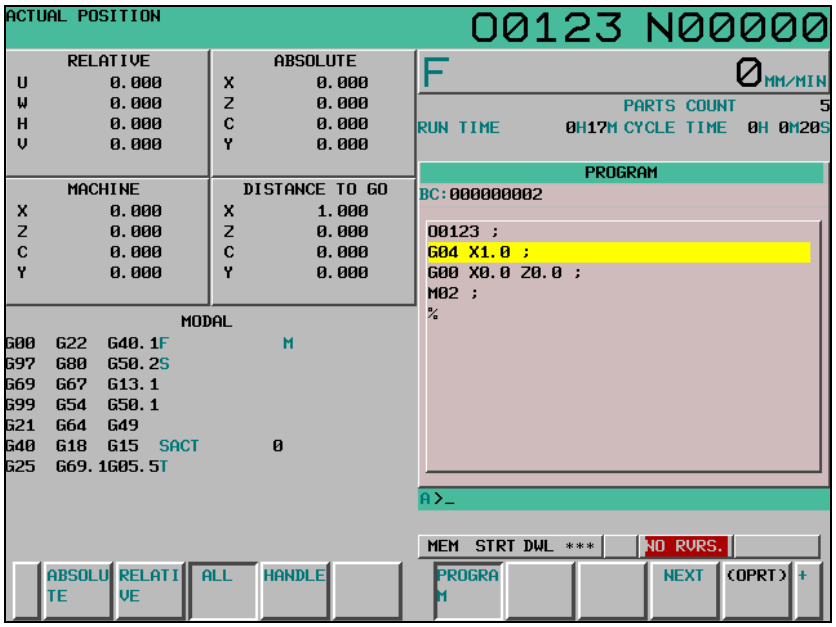


Fig. 3.5 (b) " NO RVRS." status display

Besides, when direction change prohibition signal MNCHG<F0091.1> 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.3.5(c). When the program is executed in the direction as the same as before by manual handle or direction change prohibition signal MNCHG<Fn091.1> is set to "0", the " M.H.RTR " is displayed again.
Moreover, when parameter FWD(No.6400#1) 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<Fn091.1> 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.

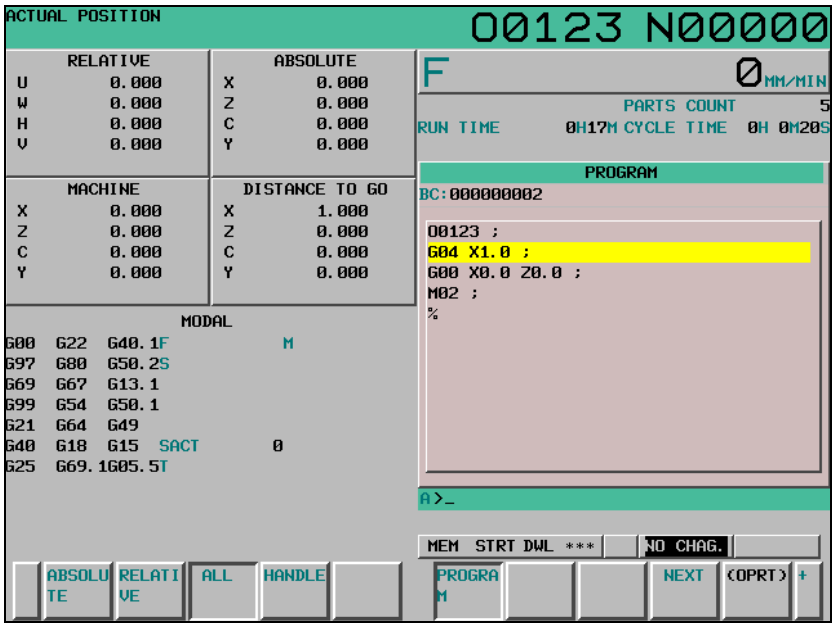


Fig. 3.5 (c) "NO.CHAG." status display

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

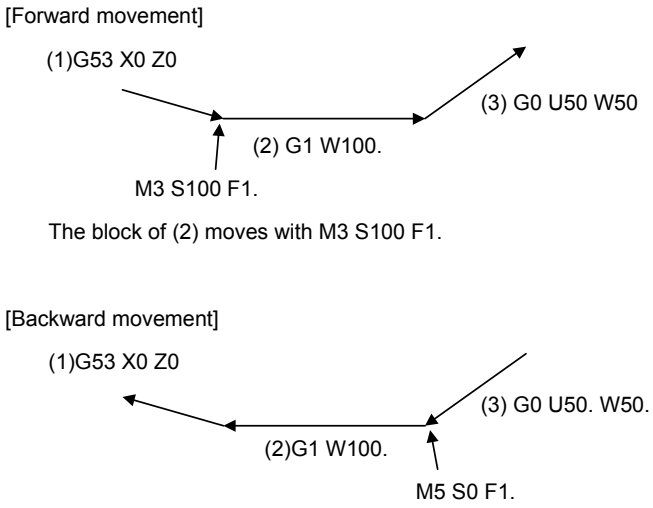
In M198 or M-code for subprogram operation by external subprogram call (parameter No.6030), the backward movement is prohibited though the forward movement is enable.

- 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".

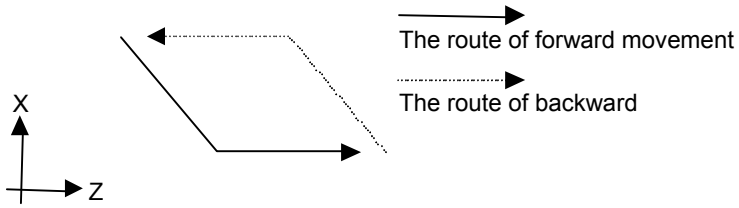
Example of executing the following programs with T series

O0001 ;
M5 S0 F0 ;
G53 X0 Z0 ; .....(1)
G1 W100 M3 S100 F1. ; .....(2)
G0 U50. W50. ; .....(3)
M2 ;



- Non linear interpolation type positioning

In the non-linear interpolation type positioning, the route is different between forward movement and backward movement.



Please use the interpolation type positioning to ward off danger. (Set the parameter LPR (No.1401#1) to "1")

When non-linear interpolation type positioning is used, inversion is prohibited as soon as any of the axes ends movement.

- Threading in forward movement

Threading 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

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.

- Simultaneous 2-path check in a 2-path system (T series (2-path control))

When the manual handle retrace function is used with more than one path simultaneously, a slight deviation in the block operation timing may result among the paths as forward and backward movement is repeated and the rotation speed of the manual handle changes. Where the blocks of different paths need to be synchronized, use the waiting M code.

- Waiting in a 2-path system (T series (2-path control))

In a 2-path system, the total of the handle pulses that are input from the beginning to the end of each block is recorded while the tool moves forward and, during backward movement, control is exerted so that the program does not proceed to the previous block until the same number of handle pulses recorded during forward movement are input. Since those handle pulses input during the in-position check are also recorded, if the handle rotation speed (i.e., axis movement speed) differs during the forward and backward movements, the time it takes to execute the in-position check changes, resulting in the total of the handle pulses recorded during forward movement being different from that recorded during backward movement. In this case, even if a block reaches its end during backward movement (the remaining travel distance displayed is 0), the program may not proceed to the previous block until the handle is rotated backward by the number of handle pulses recorded during forward movement. Note also that, because inversion is prohibited in this state, you cannot change to forward movement unless backward movement is continued allowing the program to proceed to the previous block.

- Path unit check in a 2-path system (T series (2-path control))

When checking one of the paths of a 2-path system, set a mode other than MEM for the path that is not checked. Even if 1 is set in bit 4 (HMP) of parameter No. 6400, the forward, inversion, and backward checks can be made on one path.
When the program contains the waiting M code, input the 2-path wait ignore signal NOWT<Gn063.1>.

- 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<Gn067.2> is set to "0" and handle available signal in checking mode MCHK<Gn067.3> 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/AI advanced preview control (M series)

If the checking mode signal MMOD<Gn067.2> is "1", AI contour control mode 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.

- Multi Spindle

During the backward movement, both TYPE-A and TYPE-B multi spindle control may not be operated exactly.

- Execution of measurement G-code with the speed of override 100%

When the parameter MGO(No.6400#6) is set to "1", a handle pulse is invalid and it is always executed at 100% override. When the parameter MGO(No.6400#6) is set to "0", this function is invalid and a handle pulse is valid.
In the 2-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.) of automatic tool compensation is executed, a handle pulse is invalid and it is always executed at 100% override regardless of setting the parameter MGO(No.6400#6). At the rapid traverse before measurement, handle pulse is valid.

When the parameter SKF(No.6200#7) is set to "0" and the parameter SFN(No.6207#2) is set to "0", handle pulse at G31 is invalid and it is always executed at 100% override regardless of setting the parameter MGO(No.6400#6).

When the parameter MG4(No.6400#7) is set to "1" and the software option of multistage skip is enabled and the setting of parameter from No.6202 to 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

NOTE
Multistage skip is optional function.

- Relation to another function

This function cannot coexist with the following functions

- Retrace

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6400	MG4	MGO	RVN	HMP	MC8	MC5	FWD	RPO
	MG4	MGO	RVN		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 parameter 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) Bit 6 (HST) of parameter No. 6401, which specifies whether to enable or disable status display, is set to 1.
(2) Check mode output signal MMMOD<Fn091.3> is set to 1.
1: The status is displayed if the following conditions are all satisfied:
(1) Bit 6 (HST) of parameter No. 6401, which specifies whether to enable or disable status display, is set to 1.
(2) Cycle start lamp signal STL<Fn000.5> is set to 1.
(3) Check mode input signal MMOD<Gn067.2> is set to 1.
(4) 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
6402			MWR					

[Input type] Parameter input
[Data type] Bit path

#5 MWR When the manual handle retrace function is used, for a handle operation placed in the wait state by a wait M code during backward movement:
0: Inversion is prohibited.
1: Inversion is permitted.

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	
6414	M code of group A in manual handle retrace (4)
6415	M code of group B in manual handle retrace (1)
to	
6418	M code of group B in manual handle retrace (4)
6419	M code of group C in manual handle retrace (1)
to	
6422	M code of group C in manual handle retrace (4)
6423	M code of group D in manual handle retrace (1)
to	
6426	M code of group D in manual handle retrace (4)
6427	M code of group E in manual handle retrace (1)
to	
6430	M code of group E in manual handle retrace (4)
6431	M code of group F in manual handle retrace (1)
to	
6434	M code of group F in manual handle retrace (4)
6435	M code of group G in manual handle retrace (1)
to	
6438	M code of group G in manual handle retrace (4)
6439	M code of group H in manual handle retrace (1)
to	
6442	M code of group H in manual handle retrace (4)
6443	M code of group I in manual handle retrace (1)
to	
6446	M code of group I in manual handle retrace (4)
6447	M code of group J in manual handle retrace (1)
to	
6450	M code of group J in manual handle retrace (4)
6451	M code of group K in manual handle retrace (1)
to	
6454	M code of group K in manual handle retrace (4)
6455	M code of group L in manual handle retrace (1)
to	
6458	M code of group L in manual handle retrace (4)
6459	M code of group M in manual handle retrace (1)
to	
6462	M code of group M in manual handle retrace (4)
6463	M code of group N in manual handle retrace (1)
to	
6466	M code of group N in manual handle retrace (4)

6467	M code of group O in manual handle retrace (1)
to	
6470	M code of group O in manual handle retrace (4)
6471	M code of group P in manual handle retrace (1)
to	
6474	M code of group P in manual handle retrace (4)
6475	M code of group O in manual handle retrace (1)
to	
6478	M code of group Q in manual handle retrace (4)
6479	M code of group R in manual handle retrace (1)
to	
6482	M code of group R in manual handle retrace (4)
6483	M code of group S in manual handle retrace (1)
to	
6486	M code of group S in manual handle retrace (4)
6487	M code of group T in manual handle retrace (1)
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 bit 2 (MC5) and bit 3 (MC8) of parameter No. 6400.

18060	Backward movement prohibition M code that outputs no 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.

This backward movement prohibition M code is not output to the PMC as an M code. Set an M code that is not used by an auxiliary function or macro as the backward movement prohibition M code.

18065	**Backward movement prohibition M code 1 that outputs an M code**
18066	**Backward movement prohibition M code 2 that outputs 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.

These backward movement prohibition M codes are output to the PMC as M codes. Set M codes that are not used by an auxiliary function or macro as the backward movement prohibition M codes.

Signal

Checking mode signal MMOD<Gn067.2>

[Classification] Input signal
[Function] The status of the manual handle retrace mode is changed.
[Operation] In case that the automatic operation is executed in the MEM mode when this signal is "1", the mode of the manual handle retrace becomes "ON".
In the MDI operation, the mode becomes "OFF".
Moreover, when this signal is set to "1" in the automatic operation in the MEM mode, the mode will become "ON" from the block executed next.
When this signal is set to "0" in the automatic operation, the mode will become "OFF" from the block executed next.

Handle available signal in checking mode MCHK<Gn067.3>

[Classification] Input signal
[Function] Enabling/disabling of the manual handle operation in the manual handle retrace mode is changed.
[Operation] When this signal is set to "1" in the manual handle retrace mode, the program is executed with the speed corresponding to the rotating speed of a manual steering wheel. Moreover, when the manual handle is rotated to the opposite direction, the program is executed to the opposite direction.
When this signal is set to "0", the program is executed with the speed instructed in the program to the direction of the forward movement as same as a usual automatic operation.
When this signal is changed from "1" to "0" during the program, the program is executed as well.

Forward movement prohibition signal FWSTP<Gn531.0>

[Classification] Input signal
[Function] The forward movement in the manual handle retrace mode is prohibited.

[Operation] When this signal is set to "1" in the manual handle retrace mode, the forward movement of the program is prohibited. If this signal is set to "1" during the status of direction change prohibition or the status of the backward movement prohibition, both of the forward movement and the backward movement cannot be done.

Reverse movement prohibition signal MRVM<Gn531.1>

[Classification] Input signal

[Function] The backward movement in the manual handle retrace mode is prohibited.

[Operation] When this signal is set to "1" in the manual handle retrace mode, the backward movement of the program is prohibited. The backward movement is not done and the program stops even if a manual handle is rotated to the opposite direction.

Check mode confirmation signal MMMOD<Fn091.3>

[Classification] Output signal

[Function] This signal indicates the manual handle retrace mode.

[Operation] This signal becomes "1" when the manual handle retrace mode is "ON".
When the manual handle retrace mode is turned off, this signal becomes "0".

Reverse movement signal MRVMD<Fn091.0>

[Classification] Output signal

[Function] This signal indicates that program is being backward movement.

[Operation] This signal becomes "1" in the manual handle retrace mode when the backward movement is executed.
It becomes "0" during the forward movement and the re-forward movement.

Direction change prohibition signal MRNCHG<Fn091.1>

[Classification] Output signal

[Function] This signal indicates that direction change is being prohibited.

[Operation] It becomes "1" when the direction to which the program is executed cannot be changed(The status of the direction change prohibition) in the manual handle retrace mode.
When the backward movement of the program can be done, it becomes "0".

Reverse movement prohibition signal MRVSP<Fn091.2>

[Classification] Output signal

[Function] This signal indicates that the backward movement is being prohibited.

[Operation] It becomes "1" when the backward movement can be executed no more (The status of the backward movement prohibition) from a certain block in the manual handle retrace mode.
When the backward movement of the program can be executed, it becomes "0".

Signal address

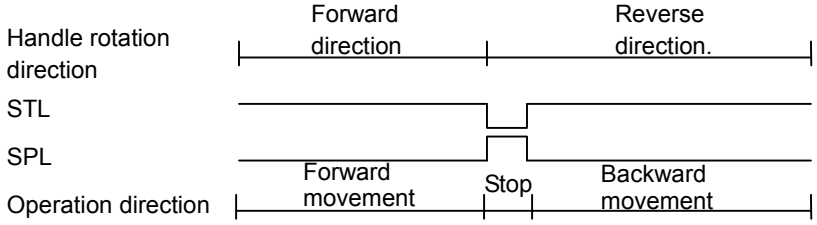
	#7	#6	#5	#4	#3	#2	#1	#0
Gn067					MCHK	MMOD		
Gn531							MRVM	FWSTP
Fn091					MMMOD	MRVSP	MNCHG	MRVMD

Notes

1) In manual handle retrace function, when check mode signal MMOD<Gn067.2> is set to "0" and handle available signal in checking mode MCHK<Gn067.3> 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.

2) When the rotate direction of the manual handle is changed, the timing chart of the cycle start lamp signal STL<Fn000.5> and the feed hold lamp signal SPL<Fn000.4> are shown in the figure below.



3) When manual handle retrace function is executed at the same time in two or more paths, some gaps might be caused in operation between each paths by repetition of forward movement and backward movement or rotational speed of manual handle. Waiting M codes are used for the synchronized block between each paths.

4 REFERENCE POSITION ESTABLISHMENT

Chapter 4, “REFERENCE POSITION ESTABLISHMENT”, consists of the following sections:

4.1 MANUAL REFERENCE POSITION RETURN 299
4.2 REFERENCE POSITION SETTING WITHOUT DOG 320
4.3 AUTOMATIC REFERENCE POSITION RETURN AND RETURN FROM THE REFERENCE POSITION 327
4.4 2ND REFERENCE POSITION RETURN / 3RD, 4TH REFERENCE POSITION RETURN 332
4.5 REFERENCE POSITION SETTING WITH MECHANICAL STOPPER 334
4.6 DISTANCE CODED LINEAR SCALE INTERFACE 338
4.7 LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL) 351
4.8 EXTENDED FUNCTION OF THE DISTANCE CODED LINEAR SCALE INTERFACE 361
4.9 CORRESPONDENCE OF ROTARY SCALE WITHOUT ROTARY DATA 368

4.1 MANUAL REFERENCE POSITION RETURN

Overview

The tool is moved in the direction specified by parameter ZMI (No. 1006#5) setting the feed axis and direction select signal to "1" during manual reference position return mode. Movement will continue until the reference position is obtained.
Manual reference position return is performed by using a grid method. The reference position is based on an electrical grid, using on one-rotation signals received from the position detector.

The following signals relate with the manual reference position return:

	Manual reference position return
Mode selection	MD1, MD2, MD4
Selection of reference position return	ZRN, MREF
Selection of axis to be moved	+J1, -J1, +J2, -J2, +J3, -J3, ...
Selection of direction to be moved	
Selection of speed to be moved	ROV1, ROV2
Deceleration signals for reference position return	*DEC1, *DEC2, *DEC3, ...
Completion signals for reference position return	ZP1, ZP2, ZP3, ...
Reference position establishment signals	ZRF1, ZRF2, ZRF3, ...

- Workpiece coordinate system setting/automatic coordinate system setting

If workpiece coordinate system setting is selected (0 is set in bit 0 (NWZ) of parameter No. 8136), the workpiece coordinate system is set when manual reference position return is completed.

If workpiece coordinate system setting is not selected (1 is set in bit 0 (NWZ) of parameter No. 8136), and if bit 0 (ZPR) of parameter No. 1201 for automatic coordinate system setting is set, the coordinate system is automatically set when manual reference position return is performed.
Parameter 1250 can be used to set the workpiece coordinate system, upon the completion of reference position return. The value set in the parameter sets the reference point of the tool holder or the tip position of the reference tool.
This operation is performed as if the following command were specified at the reference position:

G92X$\underline{\alpha}$Y$\underline{\beta}$Z$\underline{\gamma}$;

NOTE
If workpiece coordinate system setting is selected (0 is set in bit 0 (NWZ) of parameter No. 8136), automatic coordinate system setting is not performed. In this case, manual reference position return always establishes a workpiece coordinate system based on the workpiece origin offsets, specified with parameters No. 1220 to 1226.

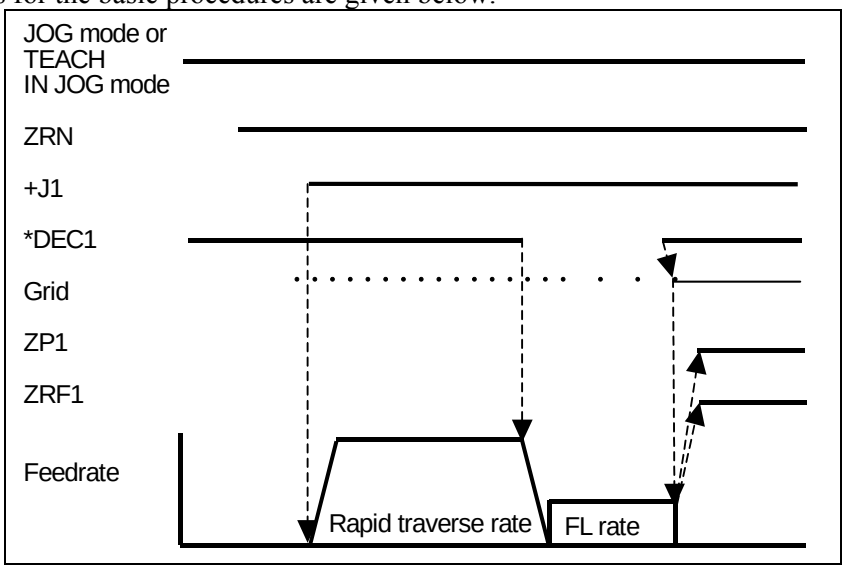
Explanation

- Basic procedure for manual reference position return

<1> Select manual continuous feed (JOG) mode, and the manual reference position return selection signal ZRN to "1".
<2> Feed a target axis toward the reference position by setting an appropriate feed axis and direction selection signal (+J1, -J1, +J2, -J2,…) to "1".
<3> While the feed axis and direction selection signal is "1", rapid traverse takes place along that axis. Although the rapid traverse override signals (ROV1, ROV2) are valid, the override is generally set to 100%.
<4> When the reference position is approached, a limit switch installed on the machine is activated, making the deceleration signal for reference position (*DEC1, *DEC2, *DEC3,…) to "0". Consequently, the feedrate is decelerated to 0, then the tool is fed at a constant low speed (reference position return FL feedrate specified by parameter (No. 1425) setting).
<5> When the deceleration signal returns to "1" again after the limit switch is passed, the tool is continues to feed, until the tool stops at the first grid point (electric grid point).
<6> Upon confirmation that the current position is at the in-position area, the reference position return end signal (ZP1, ZP2, ZP3,…) and the reference position establishment signal (ZRF1, ZRF2, ZRF3,…) turn to "1".

These steps are repeated for each axis. The number of simultaneously controlled axes is usually one, but it becomes three by setting parameter JAX (No. 1002#0).
If the feed axis direction selection signal (+J1, -J1, +J2, -J2,…) turns to "0" between step (2) and (5) , the tool is stopped at once, and reference position return is canceled. If the signal turn to "1" again, operation resumes from step (3) (rapid traverse).

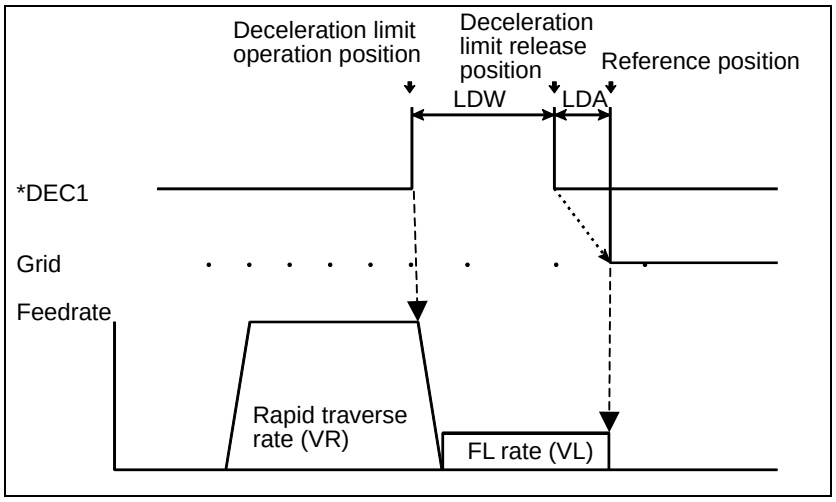
The timing charts for the basic procedures are given below.



NOTE
Grid is not a signal between PMC and CNC.

- Installation conditions for deceleration limit switch

When installing the deceleration limit switch for manual reference position return, ensure that following conditions are satisfied:



(1) LDW : Deceleration dog width [mm, inch]

$$L_{DW} > \frac{V_R\left(\frac{T_R}{2} + 30 + T_S\right) + 4V_L \times T_S}{60 \times 1000}$$

VR : Rapid traverse [mm/min, inch/min]
TR : Rapid traverse time constant [ms]
TS : Servo time constant [ms]
VL : FL speed for reference position return [mm/min, inch/min]

(2) LDA: Distance between deceleration limit switch released position and reference position
LDA = Move amount of 1/2 revolution of motor

Since the above conditions do not include the limit switch operation variations, this point must also be considered at installation.

- Servo position error and one-rotation signal

To perform the first manual reference position return after power on, the tool must be fed in manual reference position return mode, in the reference position return direction at a speed so that the servo position error exceeds the value set in parameter No. 1836. At that time, the tool must cross the grid line corresponding to a one-rotation signal from the position detector.
By the above operation, an electrical grid is established based on one-rotation signals from the position detector.

The servo position error is calculated from the following formula:

$$\text{Servo position error amount} = \frac{F \times 1000}{60} \times \frac{1}{G} \times \frac{1}{U}$$

F : Feedrate [mm/min]
G : Servo loop gain [s^{-1}]
U : Detection unit [μm]

(Example)
When the tool is fed at a feedrate F of 6000 mm/min with a servo loop gain G of 30 [s^{-1}] and a detection unit U of 1 [μm], the servo position error is calculated as follows:

$$\text{Servo position error} = \frac{6000 \times 1000}{60} \times \frac{1}{30} \times \frac{1}{1} = 3333$$

By reversing the formula above, the following formula gives the feedrate F needed to obtain a servo position error of 128, when the servo loop gain G is 30 [s^{-1}] and the detection unit U is 1 [μm]:

$$F = \frac{128 \times 60}{1000} \times 30 \quad = 230 \text{ [mm/min]}$$

Therefore, when the servo loop gain is 30 [s^{-1}], the detection unit is 1 [μm], and parameter No. 1836 is set to 128, the tool must be fed in the reference position return direction at a speed of at least 230 mm/min before completing manual reference position return.

When the one-rotation signal from the position detector is seized at the time of manual reference position return, bit 6 (PCR) of diagnosis display No. 201 is set to 1, and a grid for manual reference position return is established, enabling manual reference position return.

NOTE
This bit has no effect until the operation of the manual reference position return mode starts.

- Adjusting the reference position

There are the following reference position adjustment methods: Method by grid shift and method by reference position shift.
To shift the reference position within the distance to the first grid point, select the method by grid shift and set bit 4 (SFDx) of parameter No. 1008 to 0.
To shift the reference position beyond the distance to the first grid point, select the method by reference position shift and set bit 4 (SFDx) of parameter No. 1008 to 1.
Either of the methods for adjusting the reference position is available.

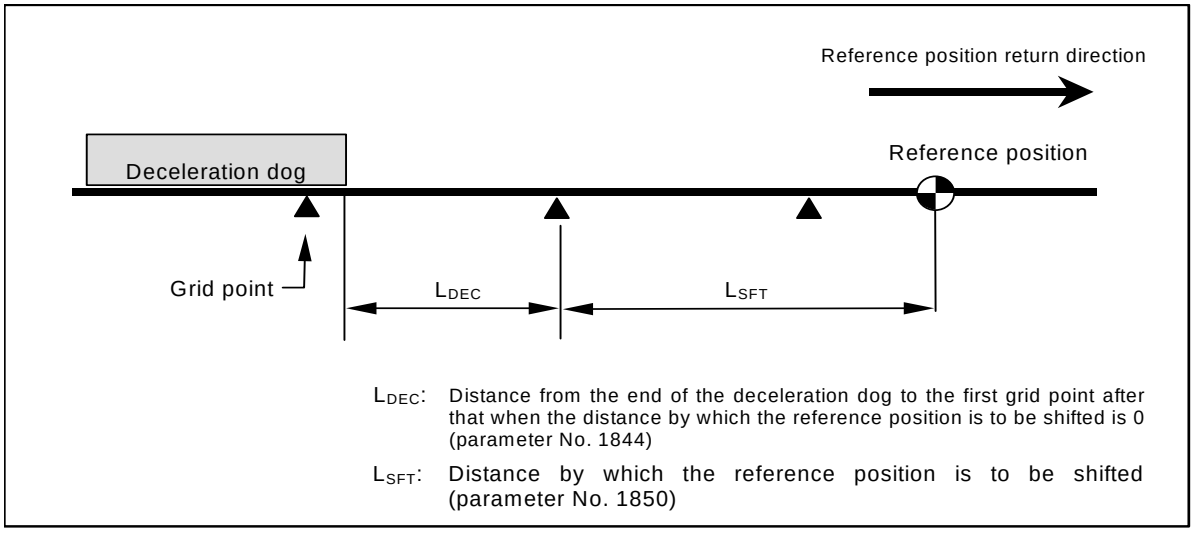
- Adjusting the reference position by grid shift

With the method for shifting the reference position by grid shift, the grid position can be shifted by the distance set in parameter No. 1850.
The distance by which the grid is to be shifted must not exceed the reference counter capacity (parameter No. 1821).
The distance between the tool and the first grid point after the tool leaves the limit switch for deceleration is indicated in diagnosis display No. 302. Furthermore, it is automatically stored in parameter No. 1844.

- Adjusting the reference position by reference position shift

With the method for shifting the reference position by reference position shift, the distance by which the reference position is to be shifted can be set in a parameter to shift it without moving the deceleration dog.

This function is enabled by setting bit 4 (SFDx) of parameter No. 1008.
The distance by which the reference position is to be shifted, shown in the figure below, can be set in parameter No. 1850 to shift the reference position.
The distance L_{DEC} for the axis along which reference position return is performed is automatically stored in parameter No. 1844. The distance L_{DEC} is indicated in diagnosis display No. 302.

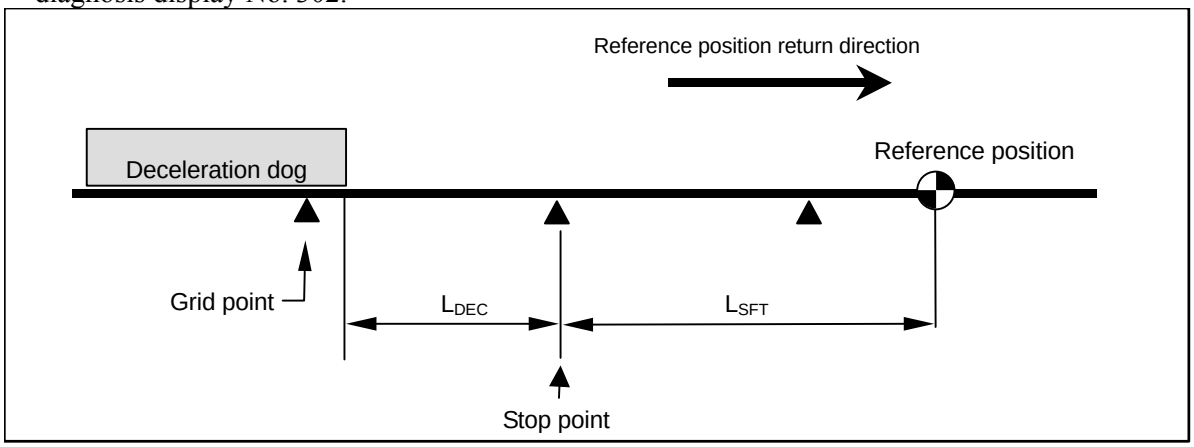


L_{DEC}: Distance from the end of the deceleration dog to the first grid point after that when the distance by which the reference position is to be shifted is 0 (parameter No. 1844)

L_{SFT}: Distance by which the reference position is to be shifted (parameter No. 1850)

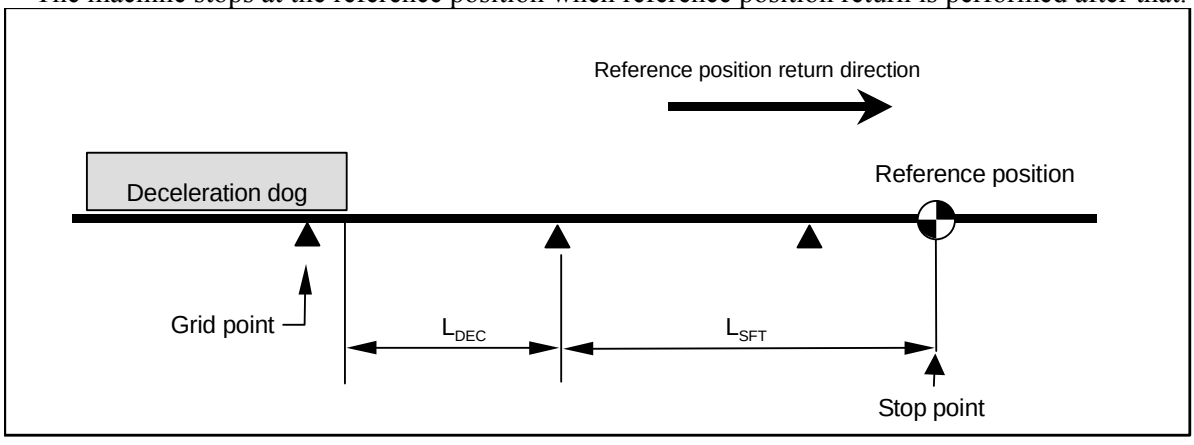
1 Set the following parameter. After setting, turn the power off, then on again.
Set bit 4 (SFDx) of parameter No. 1008 to 1.
The distance from the end of the deceleration dog to the first grid point after that (L_{DEC} (parameter No. 1844)) is 0 when the distance by which the reference position is to be shifted is 0.
Set the distance by which the reference position is to be shifted (parameter No. 1850) to 0.

2 Perform reference position return. The machine stops at the first grid point after leaving the deceleration dog.
The distance L_{DEC} is automatically set in parameter No. 1844. The distance L_{DEC} is indicated in diagnosis display No. 302.



3 Obtain the distance from the stop position to the reference position (distance by which the reference position is to be shifted (L_{SFT})) and set it in parameter No. 1850.
Then, turn the power off.
Adjusting the reference position is now completed.

4 The machine stops at the reference position when reference position return is performed after that.



⚠ CAUTION

1 Reference position shift is effective only for reference position return using grid points.
2 Install a deceleration dog so that the distance between the end of the deceleration dog and the first grid point after that is at least 1/2 of the interval between grid points.
3 When the distance by which the reference position is to be shifted is 0, the distance between the end of the deceleration dog and the first grid point after that (LDEC (parameter No. 1844)) is automatically set. Do not change the automatically set value.
4 This function cannot be used together with reference position setting without dog.
5 This function cannot be used together with the grid compensation function in feed axis synchronous control.

- Manual reference position return for a rotation axis about which the tool can rotate one or more turns

For a linear axis, when manual reference position return is performed, the relevant reference position return deceleration signal (*DEC1, *DEC2, *DEC3, ...) is set to 1. At this time, if an electric grid based on one-rotation signals from the position detector is not established, an alarm (PS0090) is issued.

For a rotation axis about which the tool can rotate one or more turns (type A) [bit 0 (ROTx) of parameter No. 1006 is set to 1 and bit 1 (ROSx) of parameter No. 1006 is set to 0], bit 1 (RTLx) of parameter No. 1007 can be set to 1 to specify that the alarm is not issued in this case.
With this setting, manual reference position return may be started with the relevant reference position return deceleration signal (*DEC1, *DEC2, *DEC3, ...) on the machine set to on. In this case, the machine moves not at the reference position return FL feedrate, but at the feedrate set in the relevant parameter.
Since the tool can rotate one or more turns about the rotation axis, it rotates one more turn. When the relevant reference position return deceleration signal (*DEC1, *DEC2, *DEC3, ...) on the machine is turned on, manual reference position return is performed in the same sequence as for normal manual reference position return.

NOTE

When bit 1 (RTLx) of parameter No. 1007 is set to 0, the relevant reference position return deceleration signal (*DEC1, *DEC2, *DEC3, ...) may be set to 1 before a grid is established. In this case, an alarm (PS0090) is issued.

For a rotation axis about which the tool can rotate up to one turn (type B) [bits 0 (ROTx) and 1 (ROSx) of parameter No. 1006 are set to 1], when the relevant reference position return deceleration signal (*DEC1, *DEC2, *DEC3, ...) is set to 1, a grid may not be established. In this case, an alarm (PS0090) is issued in the same manner as for a linear axis.

- High-speed manual reference position return after the reference position is established

After the reference position is established, the tool can be positioned at the feedrate set in the relevant parameter independently of the deceleration dog for manual reference position return. This operation is enabled by setting bit 7 (SJZ) of parameter No. 0002 and bit 3 (HJZx) of parameter No. 1005.
This operation is called high-speed manual reference position return.

After the reference position is established, setting a feed axis and direction selection signal to 1 in the manual reference position return mode positions the tool at the reference position. This positioning is performed irrespective of the direction specified by the feed axis and direction selection signal. After the completion of the positioning, the relevant reference position return end signal is set to 1.

> **⚠ CAUTION**
> For high-speed manual reference position return, a feed axis and direction selection signal may be selected in the manual reference position return mode. In this case, the tool may be positioned at the reference position irrespective of the direction specified by the feed axis and direction selection signal, depending on the current position.

If the reference position is lost, manual reference position return using a deceleration dog is performed.

Signal

Manual reference position return selection signal ZRN<Gn043.7>

[Classification] Input signal

[Function] This signal selects manual reference position return.
Manual reference position return is a kind of jog feed. Therefore, to select manual reference position return, it is required that the jog mode be selected and that the manual reference position return selection signal be set to "1".

[Operation] When the manual reference position return selection signal is set to "1", the control unit performs as described below.

- If jog feed mode is not selected, the control unit ignores the manual reference position return selection signal.
- If jog mode is selected, manual reference position return is enabled. In this case, the manual reference position return selection check signal MREF turns to "1".

> **NOTE**
> If the ZRN status changes state during jog feed, the feedrate is decelerated to 0. Then, to restart reference position return or jog feed, turn feed axis and direction selection signal to "0" then set it to "1".

Manual reference position return selection check signal MREF<Fn004.5>

[Classification] Output signal

[Function] This signal indicates that manual reference position return has been selected.

[Output cond.] This signal turns to "1" when:

- Manual reference position return has been selected.

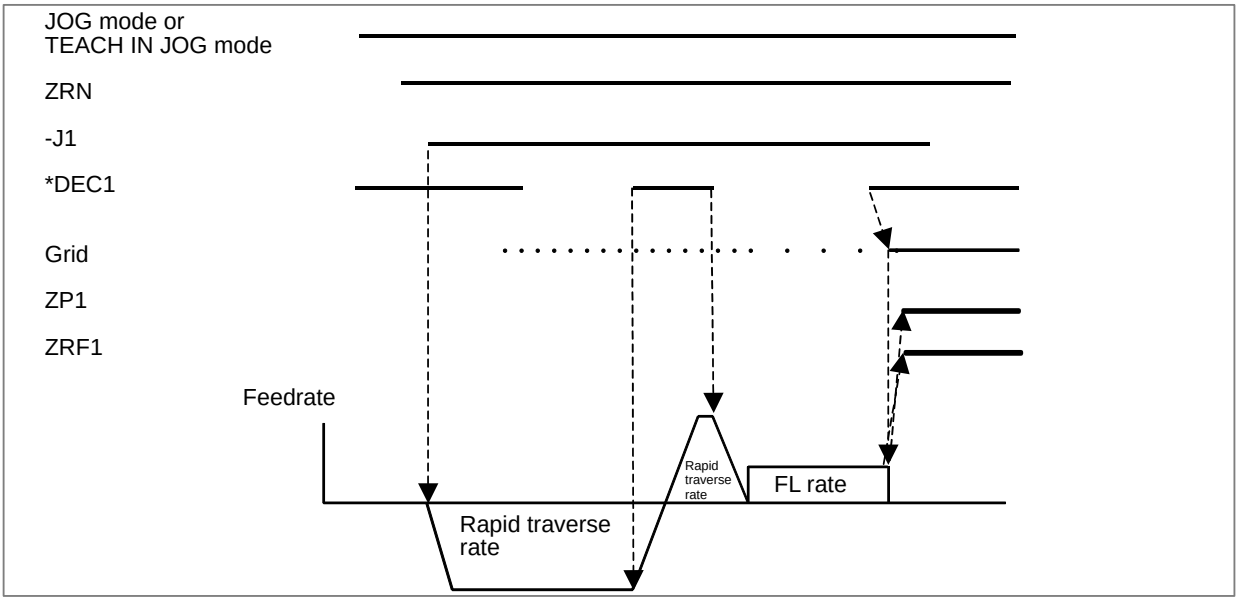
The signal turns to "0" when:

- The selection of manual reference position return has terminated.

Feed axis and direction selection signal

For details about this signal, see the explanation concerning "feed axis and direction selection signal" of Section 3.1, "JOG FEED/INCREMENTAL FEED". Only notes on use of reference position return are given, here.

NOTE
The direction of reference position return is set for each axis by parameter ZMI (No. 1006#5). If the tool is fed opposite to the preset direction in manual reference position return, while the deceleration signal is "0", the tool feeds until the signal returns to "1". The reference position return is then performed automatically in the predetermined direction.



NOTE
2 When reference position return is selected, an axis who has already completed referencing movement along that axis is disabled while the reference position return selection signal (ZRN) is "1". To perform movement again, ZRN must be set "0", and the feed axis and direction selection signal must be reset to "0" and then returned to "1" again.

Reference position return deceleration signals *DEC1 to *DEC5 <X009.0 to X009.4>

[Classification] Input signal

[Function] These signals decelerate the feedrate for manual reference position return to a low feedrate (FL).
Deceleration signals are provided for each axis. The number appended to a deceleration signal represents a controlled axis number.
*DEC<u>x</u>
x : 1:Reference position return deceleration signal for the first axis
2:Reference position return deceleration signal for the second axis
3:Reference position return deceleration signal for the third axis
:
:

[Operation] For a description of the control unit response to the deceleration signal, see the basic procedure for manual reference position return.

Bit 0 (GDC) of parameter No. 3006 can be set to 1 to use input signal G196.

NOTE
Bit 2 (XSG) of parameter No. 3008 can be set to specify whether to fix the address of the reference position return deceleration signals to X009 or use a desired X address.
When bit 2 (XSG) of parameter No. 3008 is set to 1, the X address and bit position for each reference position return deceleration signal are determined according to the setting of parameters Nos. 3013 and 3014.

Bit 5 (DEC) of parameter No. 3003 can be set to set the logic for the deceleration signals (*DEC1 to *DEC5).

Reference position return end signals ZP1 to ZP5 <Fn094.0 to Fn094.4>

[Classification] Output signal

[Function] These signals report that the tool is at the reference position on a controlled axis.
These signals are provided for each axis. The number appended to a signal represents a controlled axis number.
ZPx
x : 1 : Reference position return end signal for the first axis
2 : Reference position return end signal for the second axis
3 : Reference position return end signal for the third axis
:
:

[Output cond.] These signals are set to "1" when:
- Manual reference position returns is completed, and the axis position is in the in-position area.
- Automatic reference position return (G28) is completed, and the axis position is in the in-position area.
- Reference position return check (G27) is completed, and the axis position is in the in-position area.

These signals turn to "0"when:
- The tool has moved from the reference position.
- An emergency stop is applied.
- A servo alarm is raised.

Reference position establishment signal ZRF1 to ZRF5 <Fn120.0 to Fn120.4>

[Classification] Output signal

[Function] Notify the system that the reference position has been established.
A reference position establishment signal is provided for each axis. The number appended to each signal indicates the number of the controlled axis.
ZRFx
x : 1 : 1st-axis reference position establishment signal
2 : 2nd-axis reference position establishment signal
3 : 3rd-axis reference position establishment signal
:
:

[Output cond.] The signals are set to 1 in the following case:
- When the reference position is established after manual reference position return
- When the reference position is established using the absolute-position detector at initial power-on

When the reference position is lost, the signals are set to 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X009				*DEC5	*DEC4	*DEC3	*DEC2	*DEC1

When the parameter GDC (No.3006#0) is set to 0.

Gn196				*DEC5	*DEC4	*DEC3	*DEC2	*DEC1

When the parameter GDC (No.3006#0) is set to 1.

Gn043	ZRN							

	#7	#6	#5	#4	#3	#2	#1	#0
Fn004			MREF					
Fn094				ZP5	ZP4	ZP3	ZP2	ZP1
Fn120				ZRF5	ZRF4	ZRF3	ZRF2	ZRF1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0002	SJZ							

[Input type] Setting input
[Data type] Bit

#7 SJZ On an axis for which bit 3 (HJZx) of parameter No. 1005 is set:

0: If a reference position is not established yet, reference position return is performed with deceleration dogs.
If a reference position is already established, reference position return is performed at a parameter-set feedrate without using deceleration dogs.

1: Reference position return is performed with deceleration dogs at all times.

NOTE
SJZ is valid for an axis for which bit 3 (HJZx) of parameter No. 1005 is set to 1. When bit 1 (DLZx) of parameter No. 1005 is set to 1, however, manual reference position return after a reference position is set is performed at a parameter-set feedrate, regardless of the setting of SJZ.

	#7	#6	#5	#4	#3	#2	#1	#0
1002					AZR			JAX

[Input type] Parameter input
[Data type] Bit path

#0 JAX Number of axes controlled simultaneously in manual continuous 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 (DLZx) 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.

	#7	#6	#5	#4	#3	#2	#1	#0
1005					HJZx		DLZx	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
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.)

#1 DLZx Function for setting the reference position without dogs
0: Disabled
1: Enabled

#3 HJZx When a reference position is already set:
0: Manual reference position return is performed with deceleration dogs.
1: Manual reference position return is performed using rapid traverse without deceleration dogs, or manual reference position return is performed with deceleration dogs, depending on the setting of bit 7 (SJZ) of parameter No.0002.

When the function for setting the reference position without dogs (see the description of bit 1 (DLZx) of parameter No. 1005) is used, manual reference position return after a reference position is set is always performed at a parameter-set feedrate, regardless of the setting of HJZ.

	#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.

ROTx, ROSx Setting linear or rotation 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	Rotation 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 parameter No.1008#0(ROAx) and #2(RRLx). (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	Rotation axis (B type) (1) Inch/metric conversion, absolute coordinate values and relative coordinate values are 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 rotation 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

NOTE
For the FS0*i*-C, one of the following changes is required besides setting bit 3 (DIAx) of parameter No. 1006 so that the axis based on diameter specification achieves the specified amount of movement.
- Halve the command multiplication (the detection unit is not changed).
- Halve the detection unit and double the flexible feed gear (DMR).

For the FS0*i*-D, only if bit 3 (DIAx) of parameter No. 1006 is set, the CNC halves the specified pulse. Accordingly, the above changes are not required (when the detection unit is not changed). To halve the detection unit, double both CMR and DMR.

#5 ZMIx The direction of manual reference position return is:
0: + direction
1: - direction

	#7	#6	#5	#4	#3	#2	#1	#0
1007				GRDx			ALZx	RTLx

[Input type] Parameter input
[Data type] Bit axis

#0 RTLx When manual reference position return is performed on a rotation axis (A type) with the deceleration dog pressed before a reference position is established:
0: A movement is made at the reference position return feedrate FL.
1: Until a servo motor grid is established, a movement is not made at the reference position return feedrate FL even if the deceleration dog is pressed, but a movement is made at the rapid traverse rate.

If the deceleration dog is released after a movement at the rapid traverse rate and the deceleration dog is then pressed again and released after the rotation axis makes one revolution, reference position return operation is completed.
When this parameter is set to 0, the alarm (SW0090) "REFERENCE POSITION RETURN FAILURE" is issued if the deceleration dog is released before a servo motor grid is established.
If this alarm is issued, start manual reference position return at a position sufficiently far away from the reference position.

#1 ALZx In automatic reference position return (G28):
0: Reference position return is performed by positioning (rapid traverse).
If no reference position return is performed after the power is turned on, however, reference position return is performed using the same sequence as for manual reference position return.
1: Reference position return is performed using the same sequence as for manual reference position return.

#4 GRDx For the axis on which absolute values are detected, when correspondence between the machine position and the position by the absolute position detector is not completed, setting of the reference position without dogs is:
0: Not performed two or more times.
1: Performed two or more times.

	#7	#6	#5	#4	#3	#2	#1	#0
1008				SFDx				

[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 SFDx In reference position return based on the grid method, the reference position shift function is:
0: Disabled
1: Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
1201						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 when the workpiece coordinate system is not used (when bit 0 (NWZ) of parameter No. 8136 is 1). When the workpiece coordinate system is used, the workpiece coordinate system is established based on the workpiece origin offset (parameters No. 1220 to 1226) during a manual reference position return, regardless of the setting of this parameter.

#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 is used (when bit 0 (NWZ) of parameter No. 8136 is 0). To use the local coordinate system (G52), set bit 0 (NWZ) of parameter No. 8136 to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
1206							HZP	

[Input type] Parameter input
[Data type] Bit path

#1 HZP When a high-speed reference position return is performed, the coordinate system is:
0: Preset.
1: Not preset (FS0*i*-C-compatible specification).

NOTE
This parameter is valid when the workpiece coordinate system is not used (when bit 0 (NWZ) of parameter No. 8136 is 1) and bit 0 (ZPR) of parameter No. 1201 is 0.

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.

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 setting a coordinate system automatically.

	#7	#6	#5	#4	#3	#2	#1	#0
1401						JZR		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

#2 JZR The manual reference position return at JOG feedrate
0: Not performed
1: Performed

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.

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 JRV, bit 4 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 JRV, bit 4 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 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.

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

1420: rapid traverse rate
1423: Jog feedrate
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).

1821	Reference counter capacity 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 capacity.

As a reference counter capacity, 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.

1836	Servo error amount where reference position return is possible

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] 0 to 32767

This parameter sets a servo error used to enable reference position return.
In general, set this parameter to 0. (When 0 is set, 128 is assumed as the default.)
If, during reference position return, such a feedrate as exceeding a set value is not reached even once before the limit switch for deceleration is released (the deceleration signal (*DEC) is set to 1 again), the alarm (PS0090) "REFERENCE POSITION RETURN FAILURE" is issued.
If, during reference position return, such a feedrate as exceeding a set servo error amount is not reached even once before the limit switch for deceleration is released (the deceleration signal is set to 1 again), the alarm (PS0090) "REFERENCE POSITION RETURN FAILURE" is issued.

NOTE
When bit 0 (PLC0) of parameter No. 2000 is 1, a check is performed with a value 10 times as large as the parameter setting.
(Example)
When bit 0 (PLC0) of parameter No. 2000 is 1 and the setting is 10, if the number of servo errors is 100 or more, a reference position return is enabled.

1844	Distance to the first grid point after the deceleration dog is turned off in the case where the reference position shift amount of the reference position shift function is 0

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

Set a distance to the first grid point after the deceleration dog is turned off in the case where the reference position shift amount (No. 1850) is 0.

NOTE
This parameter is automatically set if reference position return is performed when bit 4 (SFDx) of parameter No. 1008 is set to 1, the distance to the first grid point after the deceleration dog is turned off (parameter No. 1844) is set to 0, and the reference position shift amount (parameter No. 1850) is set to 0.
Do not change an automatically set value.

1850	Grid shift and reference position shift 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 99999999

To shift the reference position, the grid can be shifted by the amount set in this parameter. Up to the maximum value counted by the reference counter can be specified as the grid shift.
In case of parameter SFDx(No.1008#4) is 0: Grid shift
In case of parameter SFDx(No.1008#4) is 1: Reference point shift

NOTE
For setting the reference position without dogs, only the grid shift function can be used.
(The reference position shift function cannot be used.)

	#7	#6	#5	#4	#3	#2	#1	#0
3003			DEC					

[Input type] Parameter input
[Data type] Bit path

#5 DEC Deceleration signal (*DEC1 to *DEC5) for reference position return
0: Deceleration is applied when the signal is 0.
1: Deceleration is applied when the signal is 1.

	#7	#6	#5	#4	#3	#2	#1	#0
3006								GDC

[Input type] Parameter input
[Data type] Bit

#0 GDC As the deceleration signal for reference position return:
0: X009 is used.
1: G196 is used. (X0009 is disabled.)

	#7	#6	#5	#4	#3	#2	#1	#0
3008						XSG		

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#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 interlock signal for each axis direction, and the tool compensation value write signal are assigned to X0000.

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 327
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 option configuration of the I/O Link, the actually usable X addresses are:
X0 to X127, X200 to X327

3014 **Bit position of an 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] Byte axis
[Valid data range] 0 to 7
Set a bit position to which the deceleration signal for reference position return (*DECn) for each axis is to be assigned.

NOTE
This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
8136								NWZ

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 NWZ Workpiece coordinate system is:
0: Used.
1: Not Used.

Alarm and message

Number	Message	Description
PS0090	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.
PS0091	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.
PS0092	ZERO RETURN CHECK (G27) ERROR	The axis specified in G27 has not returned to zero. Reprogram so that the axis returns to zero.
PS0224	ZERO RETURN NOT FINISHED	1) A reference position 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. T 2) The spindle positioning axis is commanded when the mode is not spindle positioning. Perform the spindle orientation.
PS0304	G28 IS COMMANDED WITHOUT ZERO RETURN	Although a reference position was not set, an automatic return to the reference position (G28) was commanded.
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.

Diagnosis display

302	Distance between the position at which the deceleration dog is turned off and the first grid point

[Data type] Real axis
[Unit of data] Machine unit

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Manual reference position return

4.2 REFERENCE POSITION SETTING WITHOUT DOG

Overview

This function moves each axis in the manual continuous feed mode near the reference position. It then sets the reference position in the reference position return mode without the deceleration signal for reference position return. This is done by setting the feed axis and direction select signal to "1". With this function, the machine reference position can be set at a given position without installing the limit switches for reference position return.

If the absolute-position detector is provided, the set reference position is retained after the power is turned off. In this case, when the power is turned on again, there is no need for setting the reference position again.

Explanation

- Basic Procedure for Setting the Reference Position Without DOG

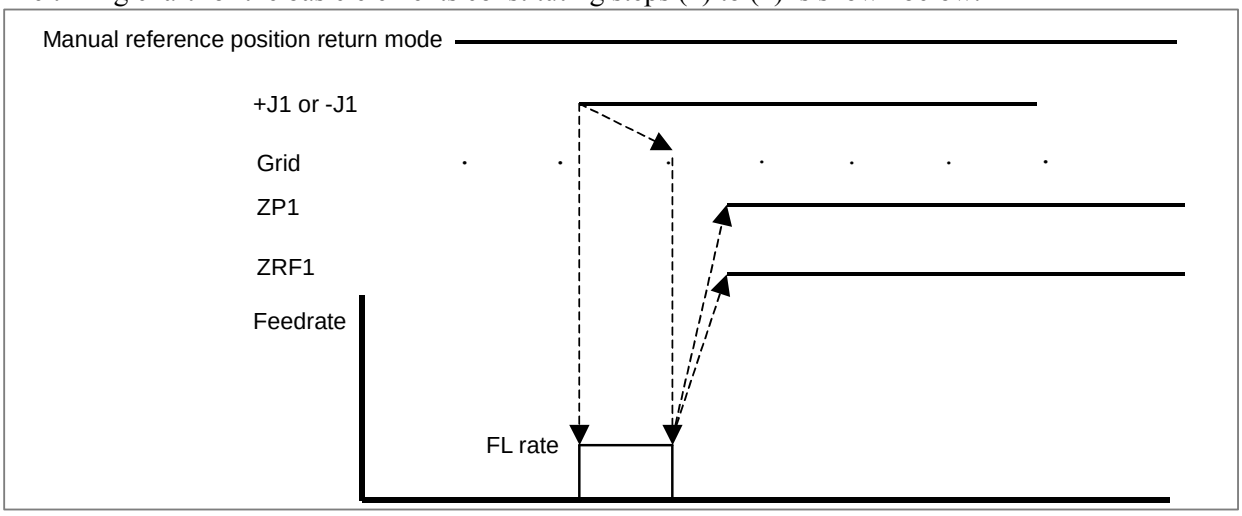
(1) Feed the tool, along the axis for which the reference position is to be set, by manual continuous feed in the reference position return direction. Stop the tool near the reference position, but do not exceed the reference position.

(2) Enter manual reference position return mode, then set 1 for the feed axis direction selection signal (for the positive or negative direction) for the axis.

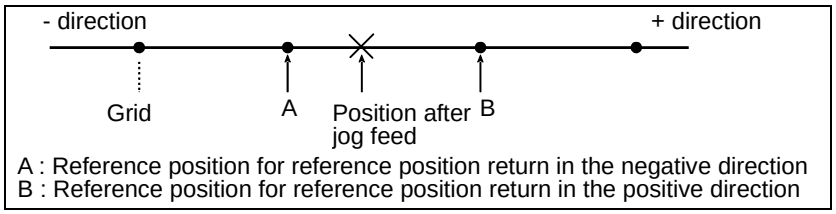
(3) The CNC positions the tool to the nearest grid point (based on one-rotation signals from the position detector) in the reference position return direction specified with parameter ZMIx (No. 1006#5). The point at which the tool is positioned becomes the reference position.

(4) The CNC checks that the tool is positioned to within the in-position area, then sets the completion signal (ZPx) for reference position return and the reference position establishment signal (ZRFx) to 1.

The timing chart for the basic elements constituting steps (2) to (4) is shown below.



The following figure shows the positional relation between the reference position and the point to which the tool is positioned by manual continuous feed.



- Servo position error and one-rotation signal

To set the reference position without DOG, when the reference position has not yet been established. The tool must be fed, in manual continuous feed mode, in the reference position return direction at such a speed that the servo position error exceeds the value set in parameter No. 1836. The tool must cross the grid line corresponding to a one-rotation signal from the position detector.
Section 4.1, "MANUAL REFERENCE POSITION RETUNR" explains how to calculate the servo position error.
When the one-rotation signal from the position detector is seized at the time of reference position setting without DOG, bit 6 (PCR) of diagnosis display No. 201 is set to 1, and a grid for manual reference position return is established, enabling setting of the reference position without DOG.

NOTE
This bit has no effect until operation in the jog mode starts during setting of the reference position without DOG.

- Grid shift

To shift the reference position, the grid can be shifted by the distance set in parameter No. 1850. The grid shift to be set in the parameter must not exceed the reference counter capacity (parameter No. 1821).
Reference position shift function can not be used.

- Manual reference position return

When the feed axis and direction selection signal is set to 1 in manual reference position return mode after the reference position has been established, the tool is positioned to the reference position regardless of the direction specified by the feed axis and direction selection signal. The completion signal for reference position return is then set to 1.

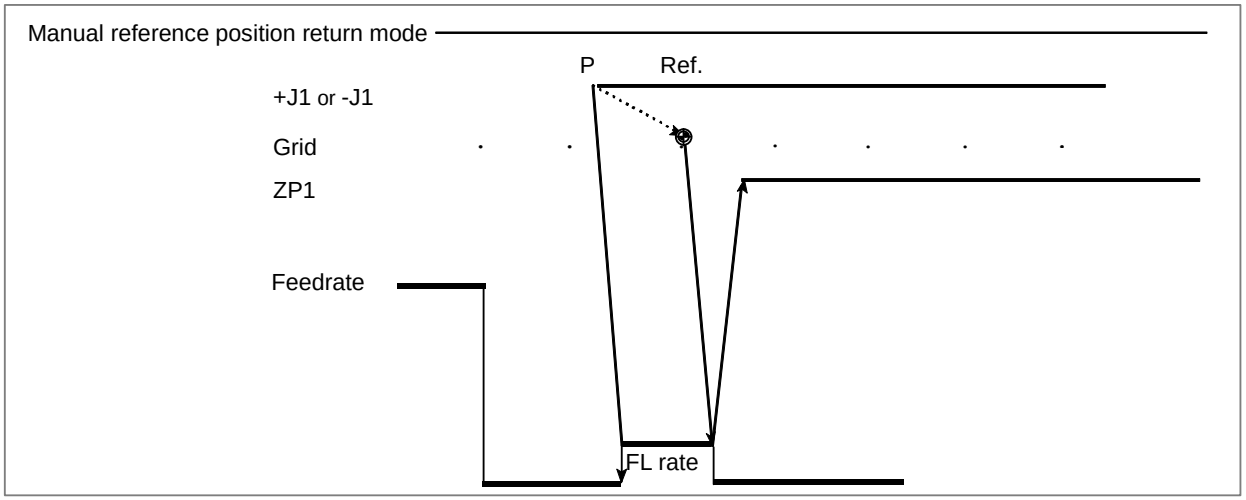
Caution

⚠ CAUTION
When a feed axis and direction selection signal is selected in the manual reference position mode, the tool may be positioned at the reference position irrespective of the direction specified by the feed axis and direction selection signal, depending on the current position.

- Setting the reference position without dog when the absolute-position detector loses the reference position

When the absolute-position detector is used (1 is set in bit 5 (APCx) of parameter No. 1815) and loses the reference position (0 is set in bit 4 (APZx) of parameter No. 1815), an alarm DS0300 occurs.

As described in "Basic Procedure for Setting the Reference Position Without DOG," feed the tool to point P in the figure below in the jog feed mode.
Then, put the machine into the manual reference position return mode and set the reference position without dog.

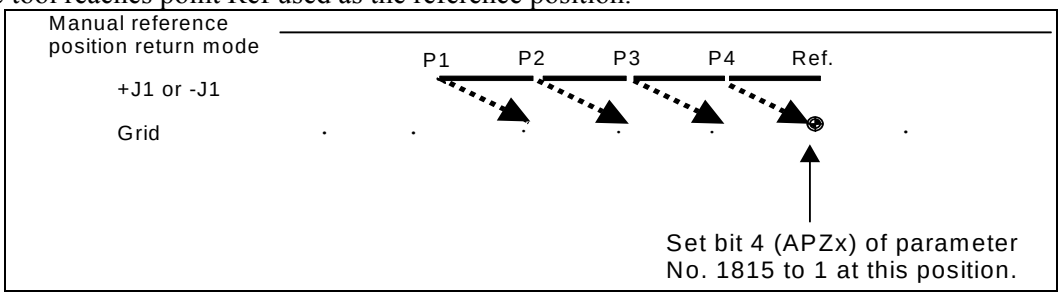


Upon the completion of setting the reference position without dog, the tool is positioned at a grid point and point Ref. in the figure is used as the reference position.
The correspondence between the machine position and reference position for the absolute-position detector has been established, and the reference position is also established. (Bit 4 (APZx) of parameter No. 1815 is set to 1.)
Then, reset the system to release the alarm (DS0300).

With the first reference position return without dog, the correspondence between the machine position and position in the absolute-position detector has been established.

Bit 4 (GRDx) of parameter No. 1007 can be set to 1 to set the reference position without dog twice or more when an alarm (DS0300) is issued due to absolute-position detection.
Bit 4 (APZx) of parameter No. 1815 remains set to 0 after the reference position is set without dog and the tool reaches the reference position.

As shown in the figure below, reference position return without dog is repeated from P2 to P3 and P4 until the tool reaches point Ref used as the reference position.



Put the system in the MDI mode at point Ref used as the reference position and set bit 4 (APZx) of parameter No. 1815 to 1.
Then, turn the power off, then on again. The correspondence between the machine position and position in the absolute-position detector has been established.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002	IDG							

[Input type] Parameter input
[Data type] Bit path

#7 IDG When the reference position is set without dogs, automatic setting of the IDGx parameter (bit 0 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
1005							DLZx	

[Input type] Parameter input
[Data type] Bit axis

#1 DLZx Function for setting the reference position without dogs
0: Disabled
1: Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
1006			ZMIx					

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#5 ZMIx The direction of manual reference position return is:
0: + direction
1: - direction

	#7	#6	#5	#4	#3	#2	#1	#0
1007				GRDx				

[Input type] Parameter input
[Data type] Bit axis

#4 GRDx For the axis on which absolute values are detected, when correspondence between the machine position and the position by the absolute position detector is not completed, setting of the reference position without dogs is:
0: Not performed two or more times.
1: Performed two or more times.

	#7	#6	#5	#4	#3	#2	#1	#0
1012								IDGx

[Input type] Parameter input
[Data type] Bit axis

#0 IDGx The function for setting the reference position again, without dogs, is:
0: Not inhibited.
1: Inhibited.

(The alarm (PS0301) is issued.)

NOTE

IDGx is enabled when the IDG parameter (bit 7 of parameter No.1002) is 1.

If the function for setting the reference position without dogs is used, and the reference position is lost in absolute position detection for a cause, the alarm (DS0300) is issued when the power is turned on again.

If the operator performs reference position return, as a result of mistakenly identifying the alarm as that requesting the operator to perform a normal reference position return, an invalid reference position may be set. To prevent such an operator error, the IDGx parameter is provided to prevent the reference position from being set again without dogs.

(1) If the IDG parameter (bit 7 of parameter No.1002) is set to 1, the IDGx parameter (bit 0 of parameter No.1012) is automatically set to 1 when the reference position is set using the function for setting the reference position without dogs. This prevents the reference position from being set again without dogs.

(2) Once the reference position is prevented from being set for an axis again, without dogs, any attempt to set the reference position for the axis without dogs results in the output of an alarm (PS0301).

(3) When the reference position must be set again without dogs, set IDGx (bit 0 of parameter No.1012) to 0 before setting the reference position.

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 setting a coordinate system automatically.

	#7	#6	#5	#4	#3	#2	#1	#0
1404							DLF	

[Input type] Parameter input
[Data type] Bit path

#1 DLF After a reference position is set, manual reference position return performed at:
0: Rapid traverse rate (parameter No.1420)
1: Manual rapid traverse rate (parameter No.1424)

NOTE
This parameter selects a feedrate for reference position return performed without dogs. This parameter also selects a feedrate when manual reference position return is performed according to bit 7 (SJZ) of parameter No.0002 using rapid traverse without deceleration dogs after a reference position is set.

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.

1821	Reference counter capacity 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 capacity.
As a reference counter capacity, 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.

1836	Servo error amount where reference position return is possible

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] 0 to 32767
This parameter sets a servo error used to enable reference position return. In general, set this parameter to 0. (When 0 is set, 128 is assumed as the default.)
If, during reference position return, such a feedrate as exceeding a set value is not reached even once before the limit switch for deceleration is released (the deceleration signal (*DEC) is set to 1 again), the alarm (PS0090) "REFERENCE POSITION RETURN FAILURE" is issued.
If, during reference position return, such a feedrate as exceeding a set servo error amount is not reached even once before the limit switch for deceleration is released (the deceleration signal is set to 1 again), the alarm (PS0090) "REFERENCE POSITION RETURN FAILURE" is issued.

NOTE
When bit 0 (PLC0) of parameter No. 2000 is 1, a check is performed with a value 10 times as large as the parameter setting.
(Example)
When bit 0 (PLC0) of parameter No. 2000 is 1 and the setting is 10, if the number of servo errors is 100 or more, a reference position return is enabled.

1850	Grid shift and reference position shift 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 99999999

To shift the reference position, the grid can be shifted by the amount set in this parameter. Up to the maximum value counted by the reference counter can be specified as the grid shift.
In case of parameter SFDx(No.1008#4) is 0: Grid shift
In case of parameter SFDx(No.1008#4) is 1: Reference point shift

NOTE
For setting the reference position without dogs, only the grid shift function can be used. (The reference position shift function cannot be used.)

Alarm and message

Number	Message	Description
PS0090	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.
PS0091	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.

Number	Message	Description
PS0224	ZERO RETURN NOT FINISHED	1) A reference position 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. T 2) The spindle positioning axis is commanded when the mode is not spindle positioning. Perform the spindle orientation.
PS0301	RESETTING OF REFERENCE RETURN IS INHIBITED	Although parameter No. 1012#0 (IDGx) 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.
PS0302	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.
PS0304	G28 IS COMMANDED WITHOUT ZERO RETURN	Although a reference position was not set, an automatic return to the reference position (G28) was commanded.
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.
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. The position control system is in error. A return to the reference position might not be performed correctly because of an abnormality inside the CNC or servo system. Restart from a manual return to the reference position.

NOTE
For reference position setting without dog, if G28 is specified before the reference position is established, an alarm (PS0304) is issued.

4.3 AUTOMATIC REFERENCE POSITION RETURN AND RETURN FROM THE REFERENCE POSITION

Explanation

- Automatic reference position return (G28)

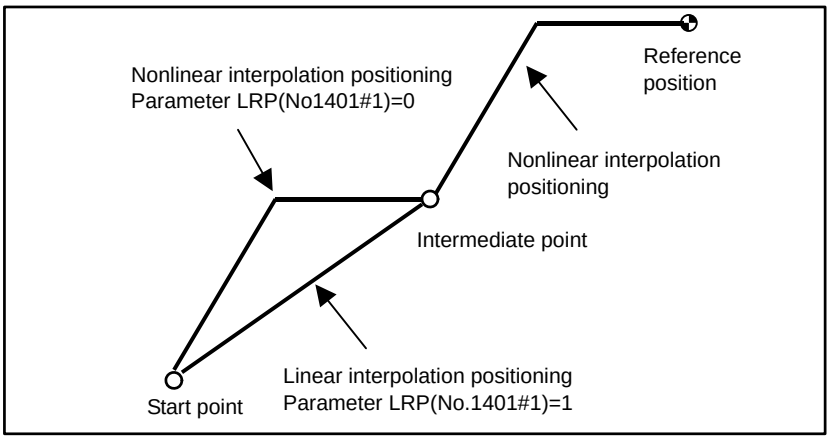
The G28 command positions the tool to the reference position, via the specified intermediate point, then sets the completion signal for reference position return (see Section 4.1, “Manual Reference Position Return”) to 1.

The reference position must be set in parameter No. 1240 (with the coordinates specified in the machine coordinate system,) before issuing the G28 command.

The tool moves to the intermediate point or reference position at the rapid traverse rate.

One of the following can be selected by LRP (bit 1 of parameter No. 1401) for the tool path to the intermediate point in automatic reference position return (G28), as with the positioning command (G00).

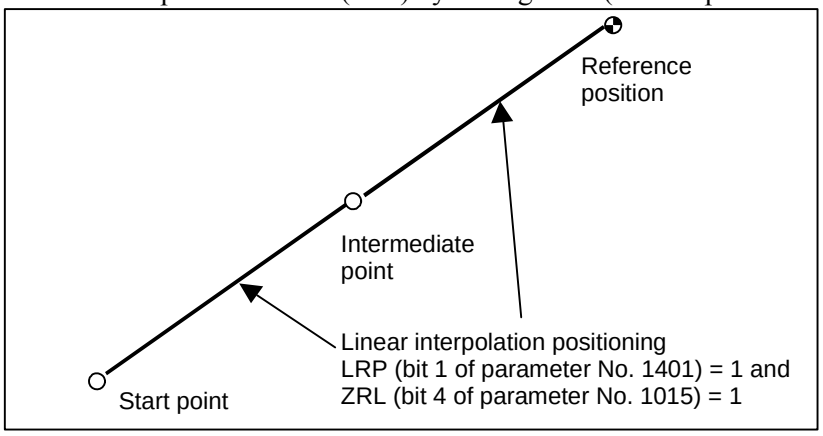
- Nonlinear interpolation positioning
  The tool is positioned with the rapid traverse rate for each axis separately. The tool path is normally straight.
- Linear interpolation positioning
  The tool is positioned within the shortest possible time at a speed that is not more than the rapid traverse rate for each axis.



In this case, nonlinear interpolation positioning is used for the tool path from the intermediate point to the reference position.

Positioning in the machine coordinate system is used for traversing from the intermediate point to the reference position by the automatic reference position return (G28) or positioning in the machine coordinate system (G53).
Generally, nonlinear interpolation positioning is used for positioning in the machine coordinate system.

When linear interpolation positioning has been selected by setting LRP (bit 1 of parameter No. 1401) to 1, linear interpolation positioning can be used for the tool path from the intermediate point to the reference position in automatic reference position return (G28) by setting ZRL (bit 4 of parameter No. 1015).



> ⚠ **CAUTION**
> If G28 is specified when the reference point is not established, nonlinear interpolation positioning is used for the tool path from the intermediate point to the reference position and the same sequence as manual reference position return is used.

- Return from reference position (G29)

Before specifying return from the reference position (G29), return to the reference position by specifying G28 or G30.

For incremental programming, the command value specifies the incremental value from the intermediate point.
Traverse from the intermediate point and specified position is performed at the speed specified by the 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 command.

After power-up, if G29 (return from the reference position) is executed even though G28 (automatic reference position return) or G30 (2nd, 3rd or 4th reference position return) has not been executed, alarm PS0305 is issued.

For the tool path in return from the reference position (G29), axis-by-axis rapid traverse rate is used for positioning to the intermediate point or the specified position. LRP (bit 1 of parameter No. 1401) can be used to specify whether nonlinear interpolation positioning or linear interpolation positioning is used for the tool path from the reference position to the intermediate point and the tool path from the intermediate point to the specified position.

NOTE
As with the positioning command (G00), positioning in the absolute coordinate system is used for traversing from the reference position to the intermediate point or traversing from the intermediate point to the specified point.
Therefore, the tool path is specified by LRP (bit 1 of parameter No. 1401).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002					AZR			

[Input type] Parameter input
[Data type] Bit path

#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.

	#7	#6	#5	#4	#3	#2	#1	#0
1006			ZMIx					

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#5 ZMIx The direction of manual reference position return is:
0: + direction
1: - direction

	#7	#6	#5	#4	#3	#2	#1	#0
1007							ALZx	

[Input type] Parameter input
[Data type] Bit axis

#1 ALZx In automatic reference position return (G28):
0: Reference position return is performed by positioning (rapid traverse).
If no reference position return is performed after the power is turned on, however, reference position return is performed using the same sequence as for manual reference position return.
1: Reference position return is performed using the same sequence as for manual reference position return.

	#7	#6	#5	#4	#3	#2	#1	#0
1015				ZRL				

[Input type] Parameter input
[Data type] Bit path

#4 ZRL When a reference position is established, the tool path from the middle point to the reference position and machine coordinate positioning (G53) in automatic reference position return (G28) and 2nd, 3rd, or 4th reference position return (G30) are based on:
0: Positioning of nonlinear interpolation type
1: Positioning of linear interpolation type

NOTE
This parameter is valid when bit 1 (LRP) of parameter No. 1401 is set to 1.

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.

	#7	#6	#5	#4	#3	#2	#1	#0
1401							LRP	

[Input type] Parameter input
[Data type] Bit path

#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.

Alarm and message

Number	Message	Description
PS0304	G28 IS COMMANDED WITHOUT ZERO RETURN	Although a reference position was not set, an automatic return to the reference position (G28) was commanded.
PS0305	INTERMEDIATE POSITION IS NOT ASSIGNED	Although a G28 (automatic return to the reference position) or G30 (return to the second, third, or fourth reference position) command was not issued after power-up, G29 (return from the reference position) was commanded.
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. The position control system is in error. A return to the reference position might not be performed correctly because of an abnormality inside the CNC or servo system. Restart from a manual return to the reference position.

Caution

⚠ CAUTION

1 If G28 is specified when the reference point is not established, traversing from the intermediate point is performed in the same sequence as with manual reference position return. In this case, the direction in which the tool moves from the intermediate point is determine by the direction of reference position return set by ZMI (bit 5 of parameter No. 1006), so the intermediate point must be a position from which reference position return can be performed.

2 If G28 is specified when the machine is locked, the reference position return complete signal is not set to 1.

3 When data is input in millimeters for an inch machine, the reference position return complete signal may be set to 1 even when there is a difference from the reference position by the minimum setting unit programmatically. This error occurs because the least input increment is smaller than the least command increment of the machine system.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Reference position return
CONNECTION MANUAL (FUNCTION) (This manual)	Linear interpolation type G28,G30,G53

4.4 2ND REFERENCE POSITION RETURN / 3RD, 4TH REFERENCE POSITION RETURN

Overview

The G30 command positions the tool to the 2nd, 3rd, or 4th reference position, via the specified intermediate point. It then sets the completion signal for 2nd, 3rd, or 4th reference position return to 1.

Before issuing the G30 command, the 2nd, 3rd, or 4th reference position must be set in parameter No. 1241, 1242, or 1243 with coordinates in the machine coordinate system.

On the tool path for return to the 2nd, 3rd, or 4th reference position (G30), the tool is moved to the intermediate point or reference position at the rapid traverse rate for each axis.
Bit 1 (LRP) of parameter No. 1401 can be set to specify whether to use nonlinear interpolation positioning or linear interpolation positioning for the tool path to the intermediate point during return to the 2nd, 3rd, or 4th reference position (G30) in the same way as for automatic reference position return (G28).
When bit 1 (LRP) of parameter No. 1401 is set to 1 to use linear interpolation positioning, bit 4 (ZRL) of parameter No. 1015 can be set to use linear interpolation positioning for the tool path from the intermediate point to the reference position during return to the 2nd, 3rd, or 4th reference position (G30).

Return to the 2nd, 3rd, or 4th reference position can be performed only after the reference position has been established.

Signal

Second reference position return end signals ZP21to ZP25 <Fn096.0to Fn096.4>
Third reference position return end signals ZP31to ZP35 <Fn098.0to Fn098.4>
Fourth reference position return end signals ZP41to ZP45 <Fn100.0to Fn100.4>

[Classification] Output signal

[Function] The second, third, and fourth reference position end signals report the tool is at the second, third, and fourth reference positions on a controlled axis.
A numeric character appended to the end of a signal represents a controlled axis number, and a numeric character immediately following ZP represents a reference position number.
Z P x y
x : 2 ..... Second reference position return
3 ..... Third reference position return
4 ..... Fourth reference position return

y : 1 ..... Return end signal for the first axis
2 ..... Return end signal for the second axis
3 ..... Return end signal for the third axis
: :

[Output cond.] These signals turn to "1" when:
- The second, third, or fourth reference position return (G30) is completed, and the current position is in the in-position area.

These signals turn to "0" when:
- The tool moved from the reference position.
- An emergency stop is applied.
- A servo alarm is raised.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn096				ZP25	ZP24	ZP23	ZP22	ZP21
Fn098				ZP35	ZP34	ZP33	ZP32	ZP31
Fn100				ZP45	ZP44	ZP43	ZP42	ZP41

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1015				ZRL				

[Input type] Parameter input
[Data type] Bit path

#4 ZRL When a reference position is established, the tool path from the middle point to the reference position and machine coordinate positioning (G53) in automatic reference position return (G28) and 2nd, 3rd, or 4th reference position return (G30) are based on:
0: Positioning of nonlinear interpolation type
1: Positioning of linear interpolation type

NOTE
This parameter is valid when bit 1 (LRP) of parameter No. 1401 is set to 1.

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.

	#7	#6	#5	#4	#3	#2	#1	#0
1401							LRP	

[Input type] Parameter input
[Data type] Bit path

#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 three-dimensional coordinate conversion, set this parameter to 1.

Alarm and message

Number	Message	Description
PS0046	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.)

Caution

⚠ CAUTION

1 If the G30 command is issued in machine lock status, the completion signal for 2nd, 3rd, or 4th reference position return is not set to 1.

2 If millimeter input is selected for an inch-system machine, the completion signal for 2nd, 3rd, or 4th reference position return may be set to 1, even when the programmed tool position deviates from the 2nd, 3rd, or 4th reference position by the least input increment. This is because the least input increment is smaller than the least command increment for the machine.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Reference position

4.5 REFERENCE POSITION SETTING WITH MECHANICAL STOPPER

Overview

This function automates the procedure of butting the tool against a mechanical stopper on an axis to set a reference position. The purpose of this function is to eliminate the variations in reference position setting that arise depending on the operator, and to minimize work required to make fine adjustments after reference position setting.

Select the axis for which the reference position is to be set, then perform cycle start. Then, the following operations are performed automatically:

1. The torque (force) of the selected axis is reduced to make the butting feedrate constant, and the tool is butted against the mechanical stopper. Then, the tool is withdrawn a parameter-set distance from the mechanical stopper.
2. Again, the torque (force) of the selected axis is reduced, and the tool is butted against the mechanical stopper. Then, the tool is withdrawn a parameter-set distance from the mechanical stopper.
3. The withdrawal point on the axis is set as the reference position.

Explanation

- Basic procedure for reference position setting with mechanical stopper

(1) First, set the parameters required for reference position setting with mechanical stopper.

Parameter ZMIx (No.1006#5)	Direction of reference position setting
Parameter No.7181	Withdrawal distance
Parameter No.7182	Reference position setting distance
Parameter No.7183	Butting feedrate 1
Parameter No.7184	Butting feedrate 2
Parameter No.7185	Withdrawal feedrate in reference position setting
Parameter No.7186	Torque limit (for 0% to 39%)
Parameter No.7187	Torque limit (for 0% to 100%)

(2) Select manual reference position return mode.

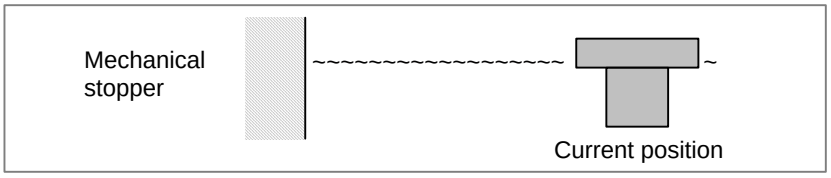
(3) By using a manual handle feed axis select signal, select the axis on which the reference position is to be set.
(4) Perform cycle start.

This starts the cycle operation for reference position setting.
During the cycle operation, the automatic operation signal OP is set to "1".

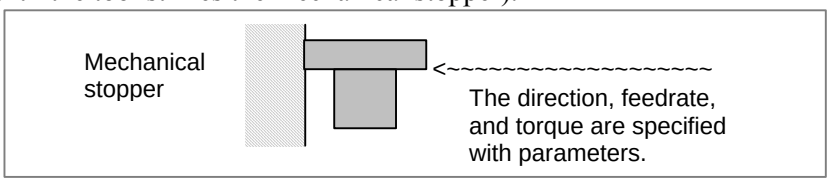
NOTE
To set the torque limit to more than 39%, set parameter No. 7187.

- Cycle operation

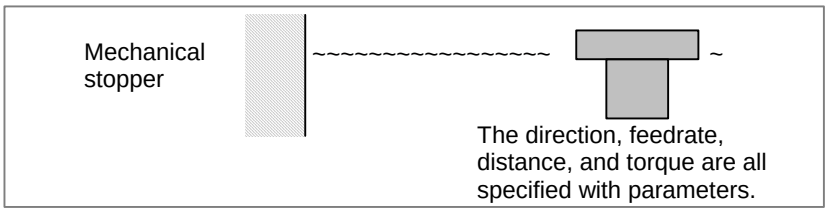
When no reference position has been set, operations (A) to (E), below, are performed automatically to set a reference position.



(A) The tool is moved along a specified axis with a limited torque until it butts against the mechanical stopper.
- The tool is moved in the direction specified with parameter ZMIx (No. 1006#5), at the feedrate specified with parameter No. 7183, at the torque specified with parameter No. 7186 or No. 7187 (until the tool strikes the mechanical stopper).

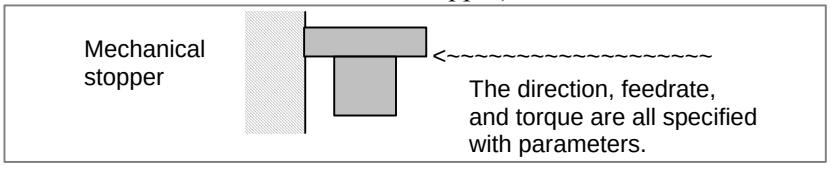


(B) After the tool strikes the mechanical stopper, the tool is withdrawn in the direction opposite to the butting direction, along the axis for a parameter-set distance.
- The tool is moved in the direction opposite to that specified with parameter ZMIx (No. 1006#5), at the feedrate specified with parameter No. 7185, for the distance specified with parameter No. 7181.

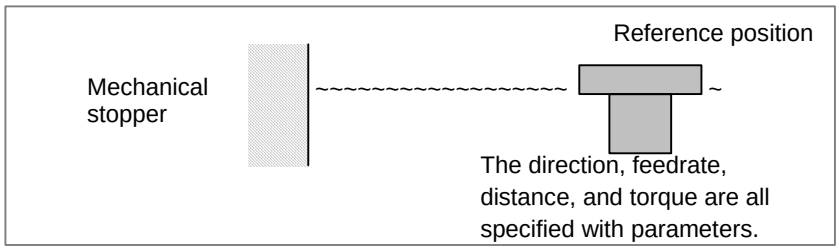


(C) Operations (D) and (E) are performed from the withdrawal point, such that the tool is butted against the mechanical stopper at a constant feedrate in reference position setting.

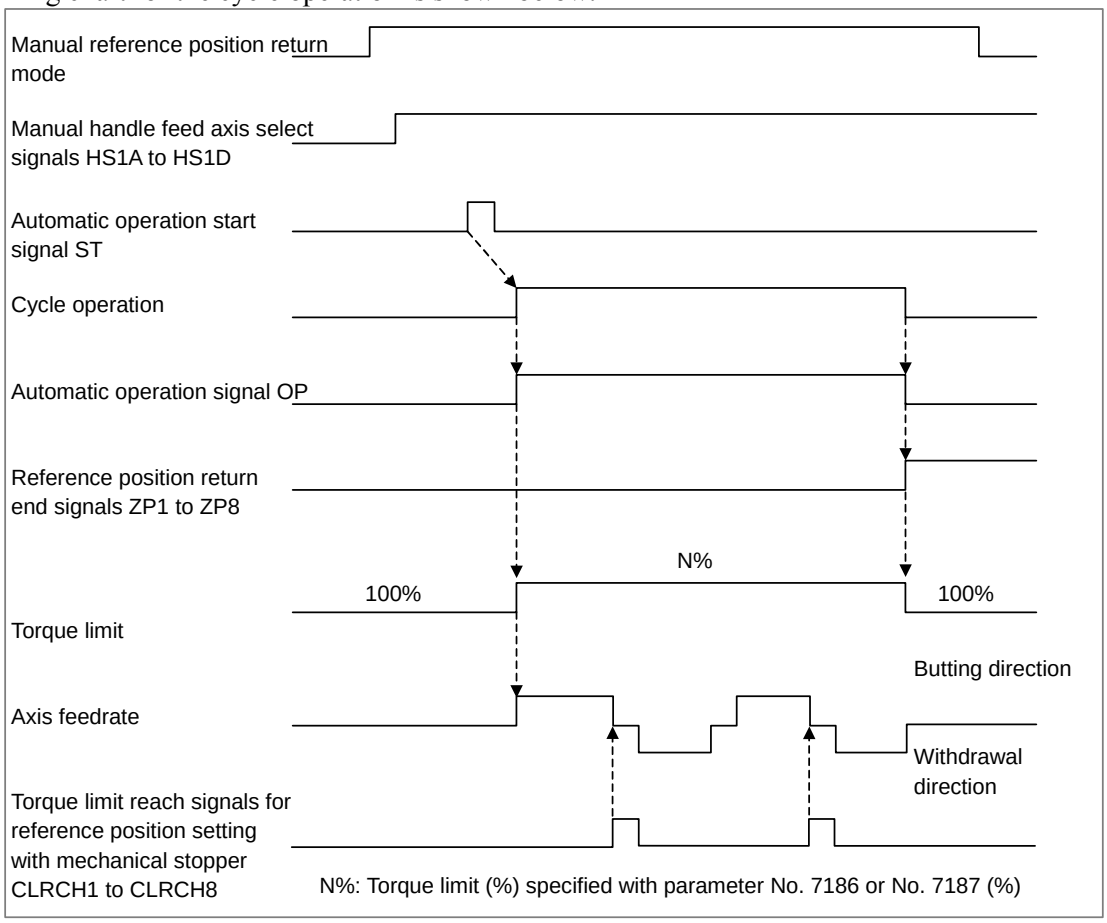
(D) The tool moves along the specified axis at a specified torque until it butts against the mechanical stopper.
- The tool moves in the direction specified with parameter ZMIx (No. 1006#5), at the feedrate specified with parameter No. 7184, at the torque specified with parameter No. 7186 or No. 7187 (until the tool strikes the mechanical stopper).



(E) After the tool strikes the mechanical stopper end on the axis, the tool is withdrawn in the direction opposite to the butting direction, along the axis for a parameter-set distance.
- The tool is moved in the direction opposite to that specified with parameter ZMIx (No. 1006#5), at the feedrate specified with parameter No. 7185, for the distance specified with parameter No. 7182.



For parameter Nos. 7183 and 7184, set the feedrates at which the tool is moved toward the mechanical stopper with a limited torque, considering the machine accuracy.
After the tool strikes the mechanical stopper, and the tool is withdrawn the distance specified with parameter No. 7182, the withdrawal point is set as the reference position on the specified axis. Then, the reference position return end signal and reference position establishment signal are set to "1".
The timing chart for the cycle operation is shown below.



- Cycle operation after the reference position has been established

When the reference position has been established, cycle operation is not performed by operations for reference position setting with a mechanical stopper. In this case, the tool is positioned to the reference position in the rapid traverse mode. After the completion of the positioning, the relevant reference position return end signal is set to "1".
This signal is not set to "1" during automatic operation.

Signal

Torque limit reach signals for reference position setting with mechanical stopper CLRCH1 to CLRCH5 <Fn180.0 to Fn180.4>

[Classification] Output signal

[Function] These signals are used to post notification of the torque limit having been reached for each corresponding axis during cycle operation for reference position setting with mechanical stopper.

[Output cond.] Each signal is set to "1" when:

- The torque limit is reached for the corresponding axis during cycle operation for reference position setting with mechanical stopper.

Each signal is set to "0" when:

- The torque limit is not reached for the corresponding axis during cycle operation for reference position setting with mechanical stopper.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn180				CLRCH5	CLRCH4	CLRCH3	CLRCH2	CLRCH1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1006			ZMIx					

[Input type] Parameter input

[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#5 ZMIx The direction of manual reference position return is:

0: + direction

1: - direction

7181	First withdrawal distance in reference position setting with mechanical stopper
7182	Second withdrawal distance in butt-type reference position setting with mechanical stopper

[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)

This parameter sets a distance an axis in each cycle operation, along which withdrawal is performed after the mechanical stopper is hit (distance from the mechanical stopper to the withdrawal point).

NOTE
Set the same direction as that set in bit 5 (ZMIx) of parameter No. 1006. Cycle operation cannot be started if the opposite direction is set.

7183	First butting feedrate in reference position setting with mechanical stopper

7184	Second butting feedrate in reference position setting with mechanical stopper

7185	Withdrawal feedrate (common to the first and second butting operations) in reference position setting with mechanical stopper

[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 feedrate used to butt against the stopper along an axis in each cycle.

7186	Torque limit value in reference position setting with mechanical stopper

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 100
This parameter sets a torque limit value. As for the setting value, 0-100 corresponds to 0% to 39%. The torque limit value is obtained by setting value ×1/255. Set the value to No.7187 to set 39% or more.

NOTE
When 0 is set in this parameter, 100% is assumed.

7187	Torque limit value in reference position setting with mechanical stopper

[Input type] Parameter input
[Data type] Word axis
[Valid data range] 0 to 255
This parameter sets a torque limit value. A value from 0 to 255 corresponds to 0% to 100%.
When this parameter is set in butt-type reference position setting, parameter No. 7186 is ignored.

NOTE
1 When 0 is set in this parameter, 100% is assumed
2 A converted setting must not exceed the rated torque.

Alarm and message

Number	Message	Description
PS0307	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.

Note

NOTE
1 Multiple axes cannot be selected simultaneously.

NOTE

2 This cycle is not executed if either of the following conditions is satisfied:
  (A) Automatic operation signal OP <F000.7> is set to "1".
  (B) The direction of reference position return (bit 5 (ZMIx) of parameter No. 1006) does not agree with the sign of a reference position setting distance (parameter No. 7181 or 7182).
3 During cycle operation, excess error, feed stop, and feed hold in the moving and stopped states are not checked.
4 During withdrawal operation, the torque limit is effective, but the tool is not butted when the limit is reached. The torque limit is released after the reference position is established.
5 When the RESET signal is input during cycle operation, operation stops, but the torque limit is not released.
6 This function cannot be used together with the grid shift function or reference position shift function.
7 This function cannot be used for angular, synchronous, Cs, spindle positioning (T series), or index table indexing axis (M series), .
8 Acceleration/deceleration is applied for this function in the same way as for manual reference position return.
9 When this function is used and the manual reference position return mode is selected, feed axis and direction selection signals (such as +J1 and -J1) must be disabled.

4.6 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.

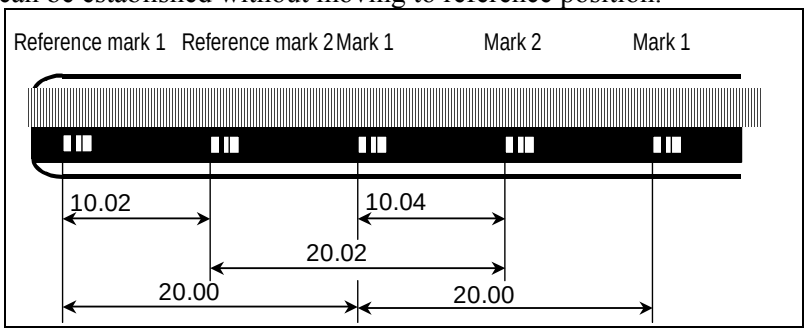


Fig.4.7(a) Example of distance coded linear scale

NOTE

To use this function requires the option of the linear scale interface with distance-coded reference marks.

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 the feed axis and 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 the reference position establishment signal (ZRF1, ZRF2, ZRF3, …) turns to "1".
(A number of reference marks is determined by the parameter No.1802#2, #1.)

The timing chart for this procedure is given below.

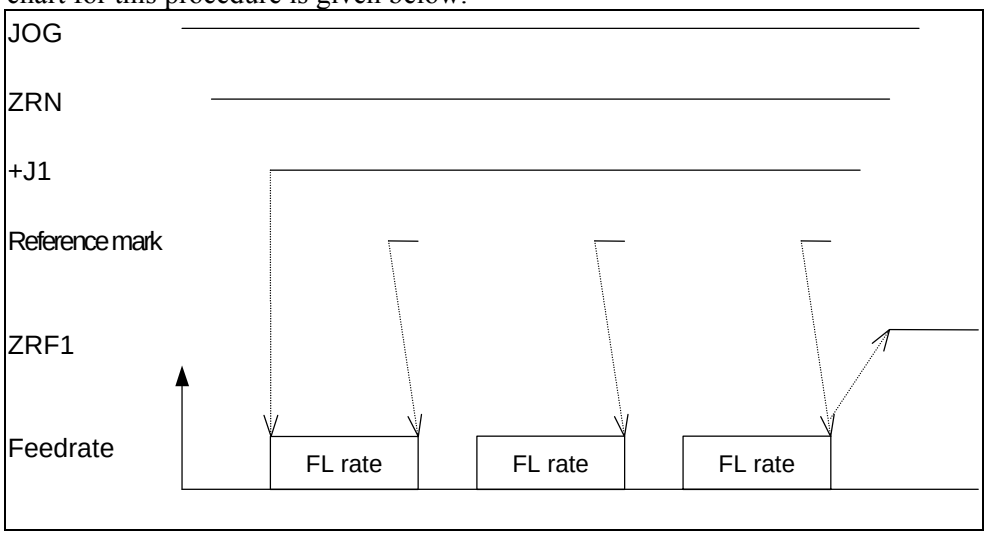


Fig.4.7(b) Timing chart for reference position establishment

- Procedure for establishing a reference position through automatic operation

If an automatic reference position returns (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 and 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.

Explanation

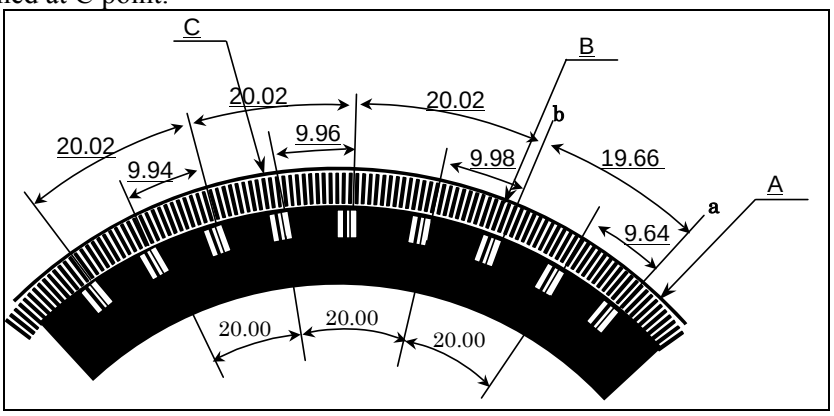
- Reference Position Return

(1) When the reference position is not established and the axis moved by turning the feed axis and 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 and 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) If an automatic reference position return (G28) is specified before a reference position is not established, a reference position is first established and, when 0 is set in bit 0 (RFS) of parameter No. 1818, the procedure is completed by moving the axis to the reference position. When 1 is set in bit 0 (RFS) of parameter No. 1818, the procedure is completed without moving the axis to the reference position.
(4) If an automatic reference position return (G28) is specified with a reference position established, the procedure is completed by moving the axis to the reference position, when 0 is set in bit 1 (RF2) of parameter No. 1818. When 1 is set in bit 1 (RF2) of parameter No. 1818, the procedure is completed without moving the axis to the middle position or reference position.

- Distance coded rotary encoder

In case of setting a rotary axis, if a parameter DCRx (No. 1815#3) 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. (parameter 1802#2(DC2) is disregarded as 0.)

- Requirements when this function is used with feed axis synchronization control axes

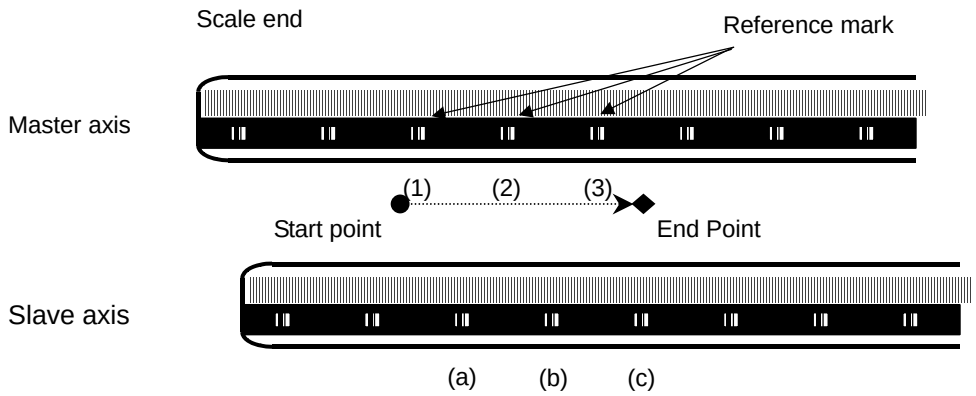
When this function is used with feed axis synchronization control axes, the distance coded linear scale used for the master axis and that used for the slave axis must have reference marks placed at identical intervals.
(Set identical values in parameter Nos. 1821 and 1882 for both the master and slave axes.)
This function cannot be enabled unless both the master and slave axes are set to use the function (1 is set in bit 2 (DCL) of parameter No. 1815).
Also, in all parameters related to this function, except parameter Nos. 1883 and 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 is issued.

NOTE
When this function is used with feed 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 to SYNC5 <Gn138>) is "1". (During establishment of a reference position, the synchronization select signal status must be maintained.)

- Reference position establishment with feed axis synchronization control axes

With feed 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 establishment signal (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 feed axis synchronization control axes, if the value of parameter No. 1883 and 1884 for both the master and slave axes is 0, the reference position is not established. Also, the reference position establishment signals (ZRF1, ZRF2, ...) are set to "0".

- PMC control axis

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.

- Arbitrary 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 is generated.

(c) During the reference position establishment operation of the angular axis, the command in the perpendicular axis is invalid in the manual reference point return.

Note

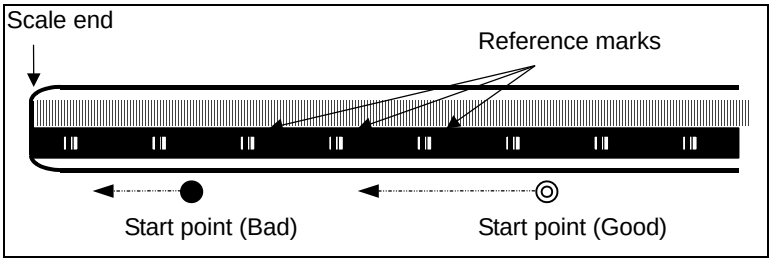
(1) In the case of the actual interval of reference marks is different from parameter setting value, the alarm DS1449 occurs.

(2) This function is disabled if any of the following conditions is satisfied:
- 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 absolute-position detection function is enabled.

(3) A difference of parameter No.1821 and No.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 parameter No. 1820 (CMR) and No.2084/2085 (flexible feed gear) to make the difference of No.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)

(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) Simple straightness compensation function (M series)
When the reference position 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 position establishment is not performed during synchronous control (T series) is activated.

(9) The reference position establishment is not performed during composite control (T series) is activated.

(10) The reference position establishment is not performed during superimposed control (T series) is activated.

Parameter

	#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 by the distance coded linear scale:
0: An absolute position is established by detecting three reference marks.
1: An absolute position is established by detecting four reference marks.

#2 DC2x When the reference position is established by the distance coded linear scale,
0: The setting of bit 1 (DC4) of parameter No. 1802 is followed.
1: An absolute position is established by detecting to 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					DCRx	DCLx	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 1 when a distance coded linear scale is used.

#2 DCLx As a separate position detector, the linear scale with a distance coded linear scale:
0: Not used.
1: Used.

NOTE
Set 1 when a distance coded linear scale is used.

#3 DCRx As the scale with absolute addressing reference marks:
0: The linear scale is used.
1: The rotary encoder is used.

NOTE
When using a rotary encoder with absolute address reference marks, set also bit 2 (DCLx) of parameter No. 1815 to 1.

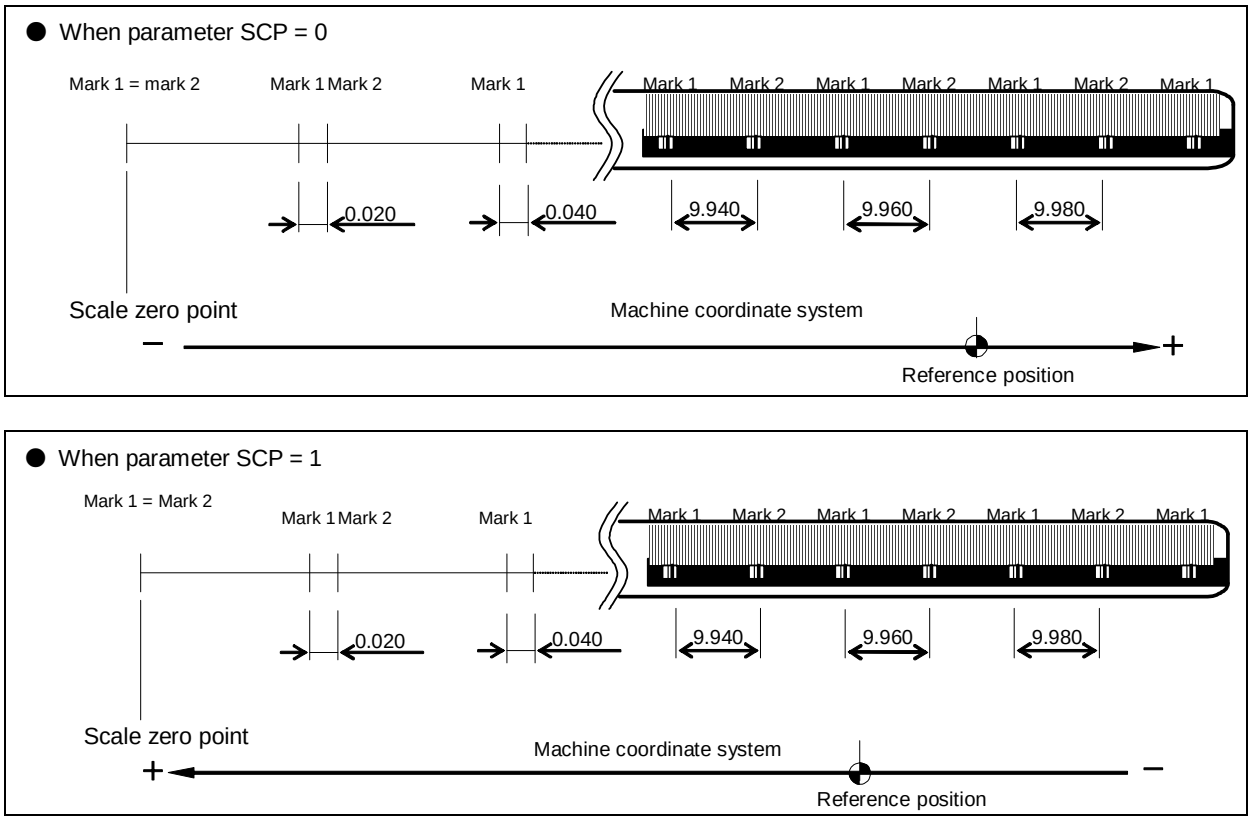
	#7	#6	#5	#4	#3	#2	#1	#0
1817				SCPx				

[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 SCPx For two-point measurement (when bit 2 (DC2) of parameter No. 1802 is set to 1), the scale zero point direction is:
0: On the minus side. (The reference position is located in the plus direction when viewed from the scale zero point.)
1: On the plus side. (The reference position is located in the minus direction when viewed from the scale zero point.)

NOTE
1 This parameter is valid when bit 2 (DC2) of parameter No. 1802 is set to 1.
2 If this parameter is set to an incorrect value, an incorrect coordinate system is established. In such a case, reverse the setting then perform reference position establishment operation again.



	#7	#6	#5	#4	#3	#2	#1	#0
1818							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 = 0) when a distance coded linear scale interface or a linear scale with distance-coded reference marks (serial) 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 already established (ZRF = 1) when a distance coded linear scale interface or a linear scale with distance-coded reference marks (serial) 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.

	#7	#6	#5	#4	#3	#2	#1	#0
1819						DATx		

[Input type] Parameter input
[Data type] Bit axis

#2 DATx When a manual reference point return is executed, an automatic setting of parameter Nos.1883 and 1884 is:
0: Not executed.
1: Executed.

The procedure of the automatic setting is following:
(1) Set a correct value to parameter Nos. 1815, 1821 and 1882.
(2) Positioning the axis to reference position by mechanical measurement by manual operation.
(3) Set parameter No.1819#2 to 1.
(4) Execute manual reference point return.
When the manual reference point return is finished, parameter Nos. 1883 and 1884 are set and No.1819#2 is turn to 0.

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.

1882	Interval of reference mark 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] 0 to 999999999
Set the interval of reference mark 2 of distance coded linear scale.

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)

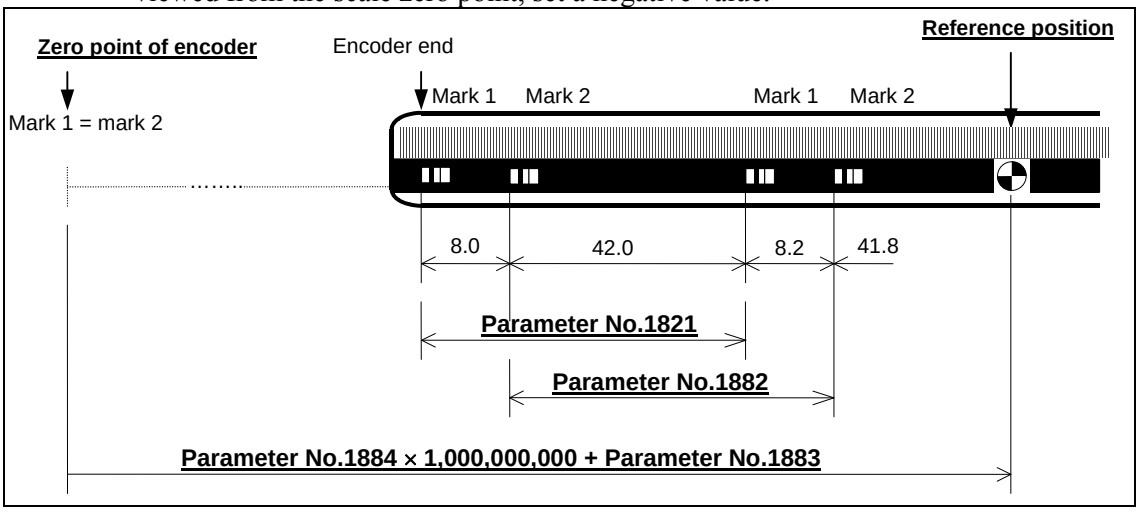
NOTE
When this parameter is set, the power must be turned off before operation is continued.

[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).

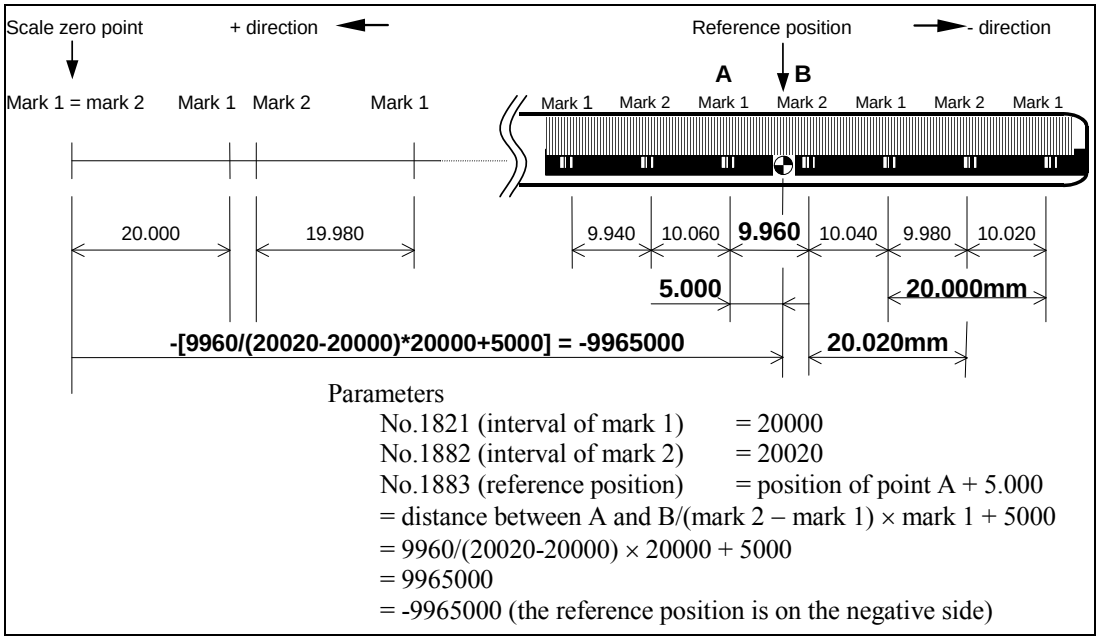
Distance from the zero point to the reference position of a linear scale
= No. 1884 × 1,000,000,000 + No. 1883

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 figure below.)
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.



[Example of parameter settings]

When an encoder as shown below is used with an IS-B, millimeter machine:



Parameters

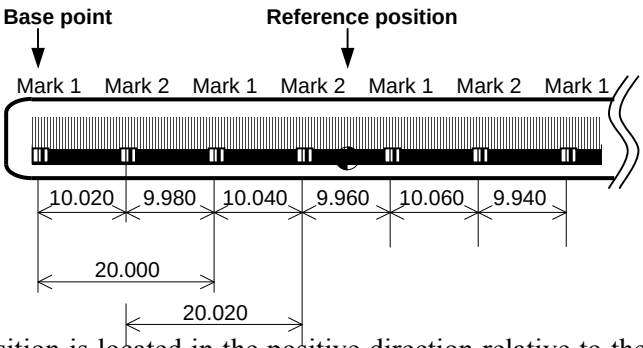
No.1821 (interval of mark 1) = 20000
No.1882 (interval of mark 2) = 20020
No.1883 (reference position) = position of point A + 5.000
= distance between A and B/(mark 2 – mark 1) × mark 1 + 5000
= 9960/(20020-20000) × 20000 + 5000
= 9965000
= -9965000 (the reference position is on the negative side)

[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.

In the case of a linear scale with the distance-coded zero point, parameters Nos. 1883 and 1884 specify the distance from the zero point to the reference position. The zero point is at an edge of the scale as shown below.



When the reference position is located in the positive direction relative to the zero point, set a positive value. When it is located in the negative direction, set a negative value. The setting procedure is as follows.

<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> Establish a reference position at an appropriate location.
(The resulting machine coordinates represent the distance from the zero point to the current position.)

<3> By jog feed or handle feed, place the machine at the accurate reference position.

<4> In parameter Nos. 1883 and 1884, set the machine coordinates converted to the detection unit (machine coordinate multiplied by CMR).

If necessary, set parameter No. 1240.

NOTE

1 Set the values of parameter Nos. 1883 and 1884 so that the distance from the scale zero point (linear scale with distance-coded reference marks) or zero point (linear scale with the distance-coded zero point) to the reference position is within the range of -999,999,999,999 to +999,999,999,999. If a value outside this range is set, an alarm PS 5325 occurs.

2 The effective area of the scale cannot extend beyond the scale zero point or zero point. Set the values so that the effective area does not extend beyond the scale zero point or zero point.

Alarm and message

Number	Message	Description
PS5220	REFERENCE POINT ADJUSTMENT MODE	In case of distance coded linear scale I/F, the reference point auto setting parameter (No.1819#2) is set to 1. Move the machine to reference position by manual operation and execute manual reference return.
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.
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 feed axis synchronization 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".
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.

Number	Message	Description
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 parameter (No.1821,1882) setting value.

Diagnosis display

3545	Measurement point 1 of the reference marks of distance coded linear scale
3546	Measurement point 2 of the reference marks of distance coded linear scale
3547	Measurement point 3 of the reference marks of distance coded linear scale
3548	Measurement point 4 of the reference marks of distance coded linear scale

[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999999999 to 999999999

3549	Internal status of the reference marks of distance coded linear scale
3550	Scale counter value of the reference marks of distance coded linear scale

[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999999999 to 999999999

3550	Scale counter value of the reference marks of distance coded linear scale (high)

[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999 to 999
Scale counter value of the reference marks of distance coded linear scale = diagnosis display (No.3551) × 1,000,000,000 + diagnosis display (No.3550)

4.7 LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL)

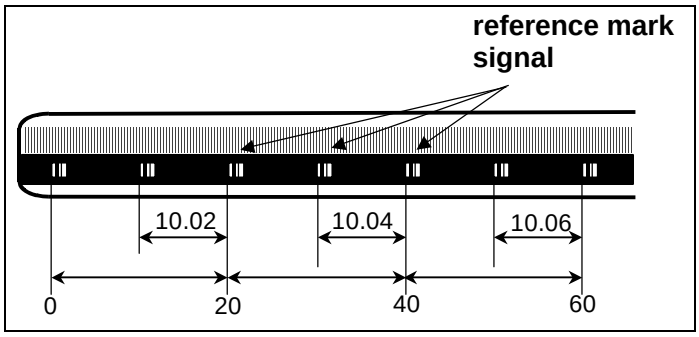
Overview

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.

NOTE
To use this function requires the option of the linear scale interface with distance-coded reference marks.

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.

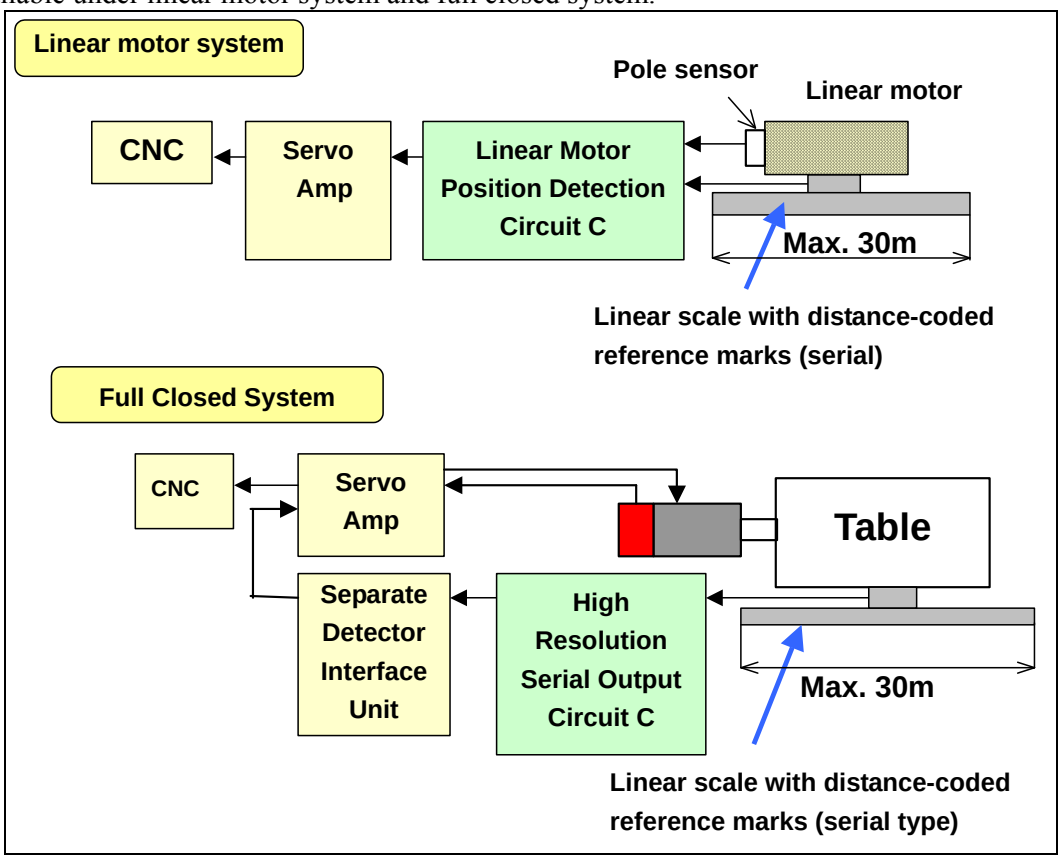


The CNC measures the interval of referenced mark by axis moving of short distance and determines the absolute position, because of the interval of each reference mark is different with regular interval.
It is not necessary that the axis is moved to the reference position for establishment of reference 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

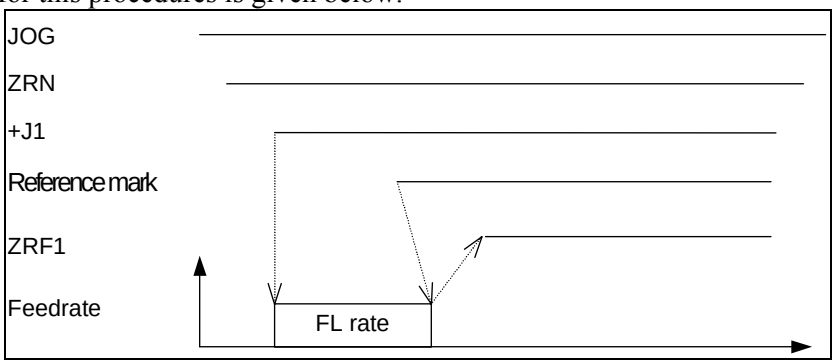
It is available under linear motor system and full closed system.



- 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 the feed axis and 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 the 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 parameter RFS No.1818#0.

- 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 and 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 parameter RFS No.1818#0.
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 parameter RF2 No.1818#1.

- Feed axis synchronization control

In case of using the feed axis synchronization control, please confirm the following items.

- When this function is used with feed 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 occurs when reference position establishment is tried.
- During operating the establishment of reference position, the state of signal for selecting synchronized axis (SYNCn<Gn138> or SYNCJn<Gn140>) should be kept.

Procedure for reference position establishment by feed 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 the reference position establishment signal (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 occurs when reference position establishment is commanded.
- When the reference position establishment of angular and perpendicular axes are tried, please set parameter 8200#2(AZR) to 0 and input signal NOZAGC <G063.5> 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 Nos.1883 and 1884 on reference position establishment (parameter DATx No.1819#2=1), please establish reference point of perpendicular axis after reference position 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.

Parameters

	#7	#6	#5	#4	#3	#2	#1	#0
1815						DCLx	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 pulse coder is not used.
1: A separate pulse coder is used.

NOTE
Set 1 when a distance coded linear scale (full-closed system) is used.

#2 DCLx As a separate position detector, the linear scale with a distance coded linear scale:
0: Not used.
1: Used.

	#7	#6	#5	#4	#3	#2	#1	#0
1818					SDCx		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 = 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 already established (ZRF = 1) when a distance coded linear scale interface or a linear scale with distance-coded reference marks (serial) 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 Linear scale with distance-coded reference marks (serial) is
0: Not used.
1: Used.

NOTE
1 When this parameter is set, the power must be turned off before operation is continued.
2 Please set parameter 1815#1(SDC) to 1 when the full closed system.

	#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 parameter No. 1883 and No. 1884 at manual reference position return time is:

0: Not performed.
1: Performed.

The procedures of the automatic setting are following:

(1) Set a correct value to parameter Nos. 1815, 1821 and 1882.
(2) Positioning the axis to reference position by mechanical measurement by manual operation.
(3) Set parameter No.1819#2 to 1.
(4) Execute manual reference point return.
When the manual reference point return is finished, parameter Nos. 1883 and 1884 are set and No.1819#2 is turn to 0.

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)

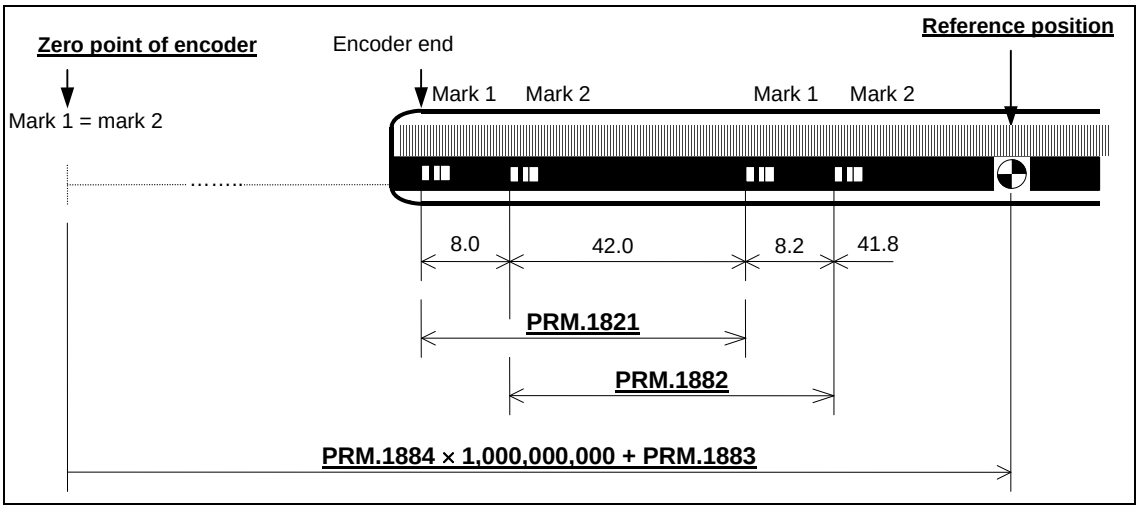
NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] -999 to 999

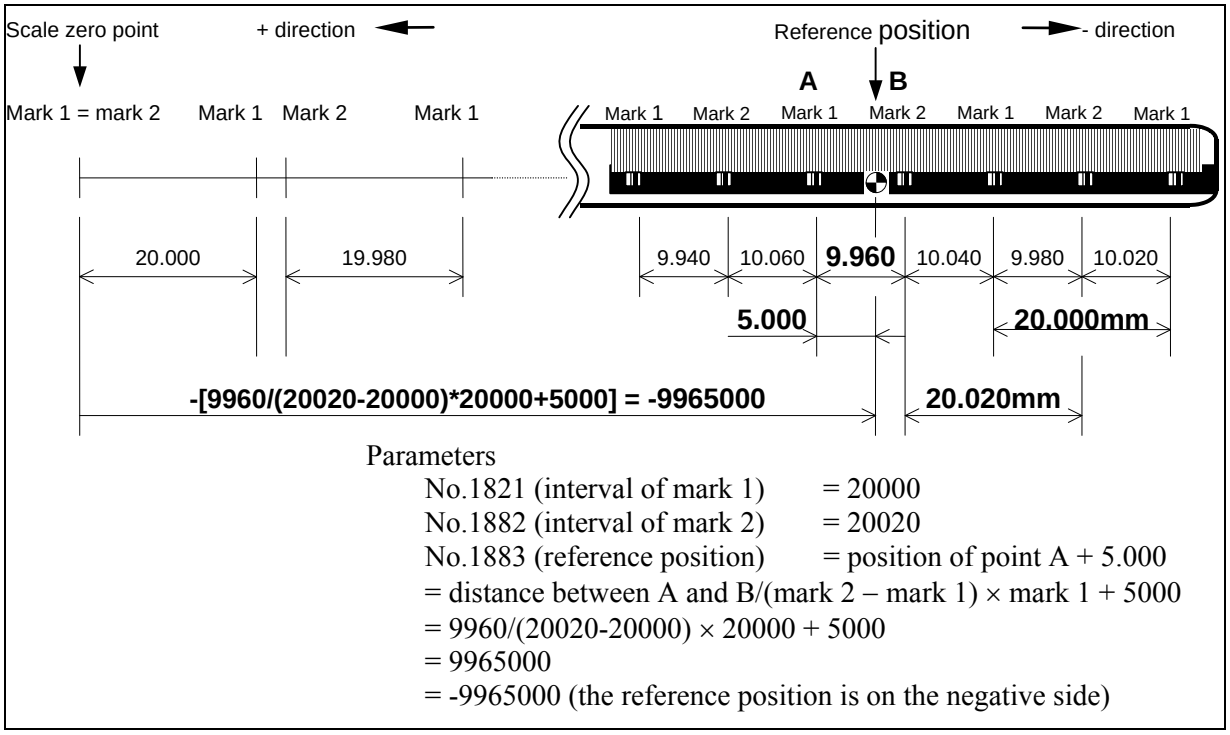
When a linear scale with absolute address reference marks is used, set the distance from the scale zero point to reference position in parameter Nos. 1883 and 1884).

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 figure below.)
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.



[Example of parameter settings] When an encoder as shown below 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)

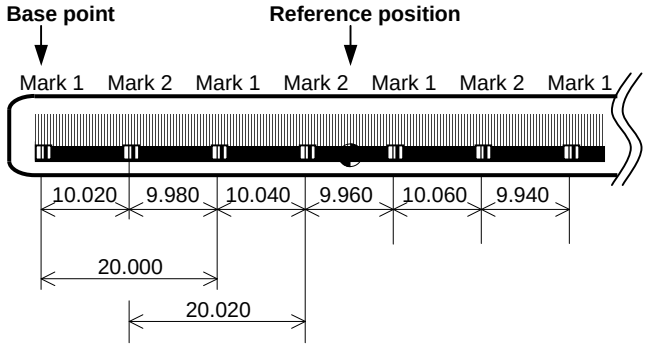
[Setting parameter No. 1883]

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 below.



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 No. 1883 and No. 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 (PS 5325) 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.

14010	Allowable move amount of FL feedrate on establishment of reference point(linear scale with distance-coded reference marks (serial))

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 999999999

On the linear scale with distance-coded reference marks (serial), the amount of movement on FL-speed during establishment of reference point is set.

If the reference point is not established despite the amount of FL-speed movement exceeds this parameter, alarm DS0017 (SERIAL DCL: REF-POS ESTABLISH ERR) occurs.
When this parameter is set to 0, the setting of allowable move amount of FL feedrate on establishment of reference point (linear scale with distance-coded reference marks (serial)) is ineffective.

NOTE
1 Set this parameter to both master axis and slave axis when the reference position is established on the feed axis synchronization control axes.
2 On angular axis control, this parameter setting of perpendicular axis is ignored during the reference position establishment of the angular axis.

Alarm and message

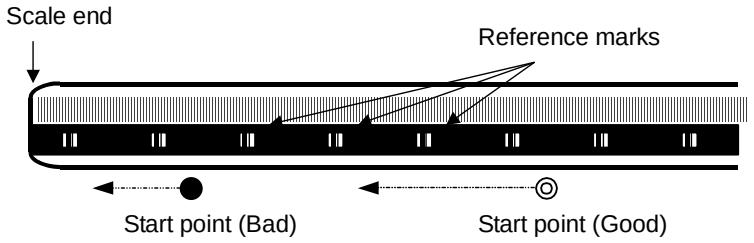
No.	Message	Description
PS0372	REFERENCE RETURN INCOMPLETE	An attempt was made to perform an automatic return to the reference position on the orthogonal axis before the completion of a return to the reference position on the angular axis. However, this attempt failed because a manual return to the reference 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 orthogonal axis.
PS0376	SERIAL DCL: ILLEGAL PARAMETER	1. When Parameter No.1815#1 is set to 1, parameter No.2002#3 is set to 0 2. The absolute-position detection function is enabled. (Parameter No.1815#5 is set to 1.)
PS5220	REFERENCE POINT ADJUSTMENT MODE	In case of distance coded linear scale I/F, the reference point auto setting parameter (No.1819#2) is set to 1. Move the machine to reference position by manual operation and execute manual reference return.
DS0016	SERIAL DCL:FOLLOW-UP ERROR	1. The value of parameter No.1883,No.1884 is out of range. 2. The amount of follow-up is more than 2147483647 or less than -2147483648 on detection unit. To reduce the follow-up amount, please adjust the point that we begin follow-up.
DS0017	SERIAL DCL:REF-POS ESTABLISH ERR	The amount of movement for establish reference position was exceeded the amount of parameter No.14010. Please try again or change parameter No.14010 larger.
DS0018	SERIAL DCL:MISMATCH (SSYNC CTRL)	Master/slave axes of feed axis synchronization control, one of them is the linear scale with distance-coded reference marks (serial) and the other of them is not the linear scale with distance-coded reference marks (serial). Please establish reference position with the input signal SYNCn<G138>, SYNCJn<G140> or parameter SCA No.8304#5 setting to 0.
DS0019	SERIAL DCL:MISMATCH(ANGL-AXIS)	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.

No.	Message	Description
DS0020	REFERENCE RETURN INCOMPLETE	An attempt was made to perform an automatic return to the reference position on the orthogonal 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 orthogonal axis.
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.
SV0382	COUNT MISS(EXT)	A count error occurred on the separate detector.
SV0383	PULSE MISS(EXT)	A pulse error occurred on the separate detector.

⚠ CAUTION

1 When the Linear scale with distance-coded reference marks (serial) is used, please set parameter SDCx No.1818#3 to 1.

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 Simple straightness compensation function (M series)
When the reference position establishment of moving axis is executed after the establishment of compensation axis, the compensation axis is moved by the simple straightness compensation amount when the reference point of moving axis is established.

4 It is not available to use this function and the temporary absolute coordinate setting together.

5 This function cannot be used together with synchronous control, composite control, and superimposed control (T series).

4.8 EXTENDED FUNCTION OF THE DISTANCE CODED LINEAR SCALE INTERFACE

Overview

The distance coded linear scale interface has reference marks at intervals that change at a constant rate. By determining the reference mark interval, the corresponding absolute position can be deduced. When a G00 command or a move command in jog feed is specified for an axis for which the distance coded linear scale interface is used, this function establishes the reference position by measuring the reference mark intervals automatically. Therefore, after CNC power-up, the reference position can be established without performing reference position return operation.

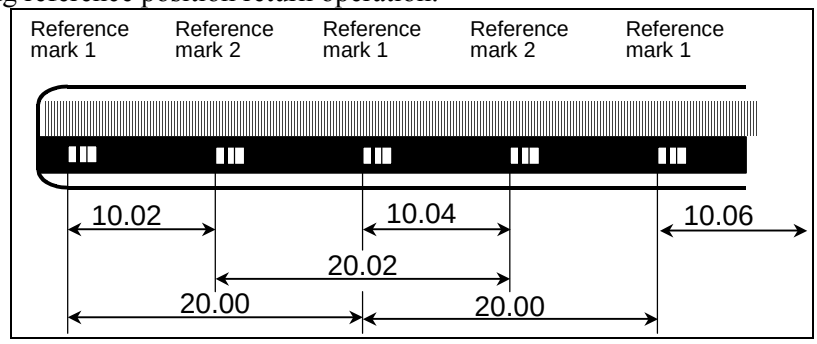


Fig. 4.8 (a) Sample distance coded linear scale interface

NOTE
The extended function of the linear scale with distance-coded reference marks is optional. To use this function, the interface option of the distance coded linear scale interface is also required.

4.8.1 Reference Position Established by the G00 Command

Explanation

- Activation conditions

When the following conditions are satisfied, reference position establishment operation is performed automatically:

<1> A G00 command is specified to cause a movement along an axis for which no reference position has been established.

<2> The movement direction in <1> above matches the reference position return direction set by bit 5 (ZMI) of parameter No. 1006.

<3> The specified axis is not in the following modes:

- Coordinate system rotation
- Mirror image (mirror image by signal or setting)
- Programmable mirror image
- Scaling (M series)
- AI contour control mode/AI advanced preview control (M series)
- Mirror image of facing tool posts (T series)

NOTE
If all the above conditions are not satisfied, the reference position establishment operation is not performed, and normal G00 command operation is performed.

- Operation

The reference position establishment procedure is explained below.

<1> The tool is fed along a specified axis at the reference position return FL feedrate (parameter No. 1425).
<2> Upon detection of a reference mark on the scale, the tool is stopped briefly then fed at the reference position return FL feedrate.
<3> Step <2> above is repeated until two, three, or four reference marks are detected on the scale. Then, the reference position is established, and the reference position establishment signal (ZRF1, ZRF2, ZRF3, etc.) is set to "1".
<4> The tool is fed to a specified end point at a rapid traverse rate.

A time chart for the above procedure is shown below.

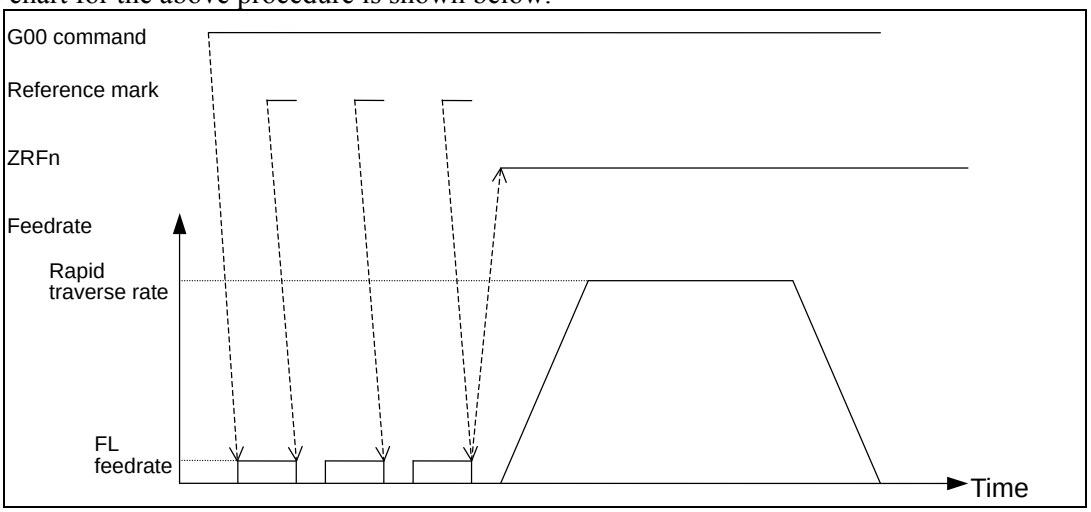


Fig. 4.8.1 (a) Time chart for reference position establishment (G00)

The specifications for the steps of detecting reference marks and establishing the reference position (steps <1> to <3> above) are the same as for the conventional distance coded linear scale interface. The restrictions are also the same.
For details, refer to "Specifications of Linear Scale with Absolute Addressing Reference Marks."

- Tool path

The tool path in the G00 command is explained below.

(1) When no axis requires the reference position establishment When the reference position has already been established for all specified axes, the reference position establishment operation is not performed.
For example, suppose that the reference position is already established for the X-, Y-, and Z-axes, and that G00 Xxx Yyy Zzz; is specified. Then, normal rapid traverse operation takes place. The tool path follows the setting in bit 1 (LRP) of parameter No. 1401.

(2) When all axes require the reference position establishment operation
Suppose that the reference position is not established for the X-, Y-, and Z-axes and that G00 Xxx Yyy Zzz; is specified. The operation in this case is shown in the figure below.
Operation 1 in the figure establishes the reference position. During the establishment operation, the tool path is always of the non-linear interpolation type regardless of the setting in bit 1 (LRP) of parameter No. 1401.
Operation 2 performs positioning to a specified end point. During this operation, the tool path follows the setting in bit 1 (LRP) of parameter No. 1401.
After operation 1 is completed for all axes, operation 2 starts.

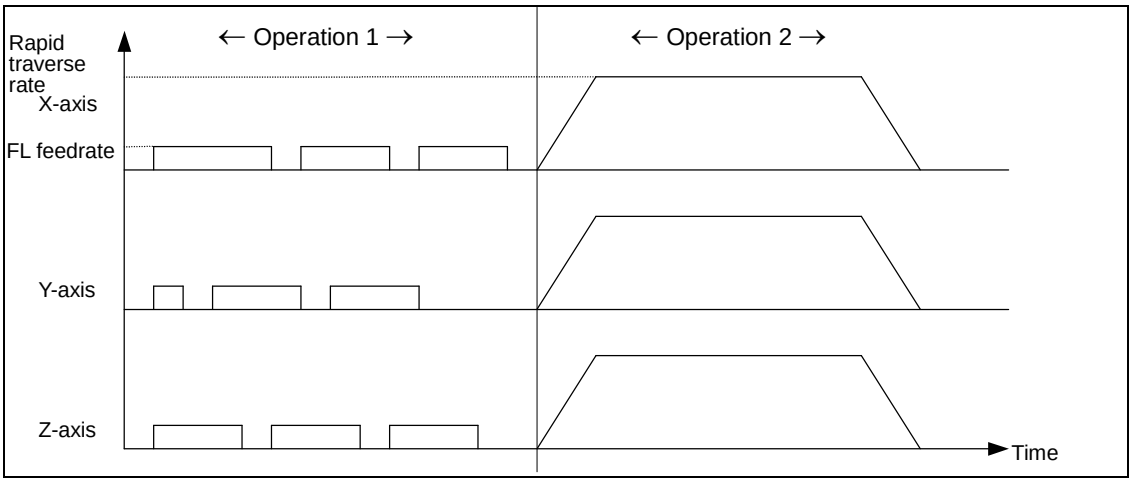


Fig. 4.8.1 (b) When the reference position is established for all axes

(3) When some axes require the reference position establishment operation and others do not require the establishment operation

For example, suppose that the reference position is already established for the X-axis and that the reference position is not yet established for the Y- and Z-axes. Also suppose that G00 Xxx Yyy Zzz; is specified. The operation in this case is shown in the figure below.

In operation 1 in the figure, movement to a specified position is made along the X-axis for which the reference position is already established. For the Y- and Z-axes for which no reference position is established, the reference position is established. During the establishment operation, the tool path is always of the non-linear interpolation type regardless of the setting in bit 1 (LRP) of parameter No. 1401.

In operation 2, positioning to a specified end point is performed along the Y- and Z-axes. The tool path along the Y- and Z-axes then follows the setting in bit 1 (LRP) of parameter No. 1401. Because positioning to the specified position is already made along the X-axis, no movement is made along the X-axis.

After operation 1 is completed for all axes, operation 2 starts.

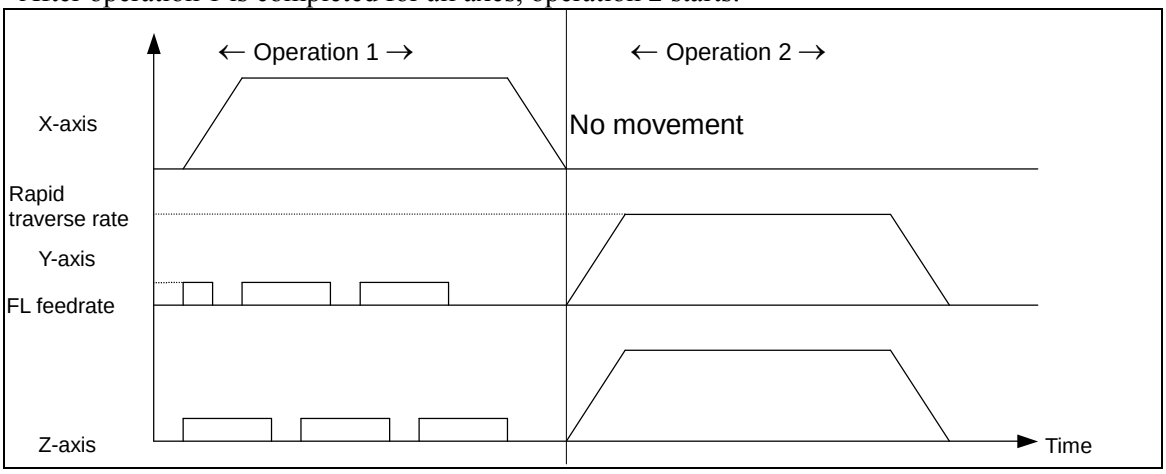


Fig. 4.8.1 (c) When an axis does not require the reference position establishment operation and others require the establishment operation

- Absolute command and incremental command

After the reference position has been established, positioning to a specified end point is performed. This operation is explained below.

(1) When an absolute command is specified
Movement to the end point in the new coordinate system that has been established in step <3> is made. For example, when G90 G00 Xxx.Yyy.; is specified, the tool moves as indicated with the bold line in the figure below. Note that, however, the figure shows the positional relationship among points and that the intermediate tool path is not always of the linear interpolation type.

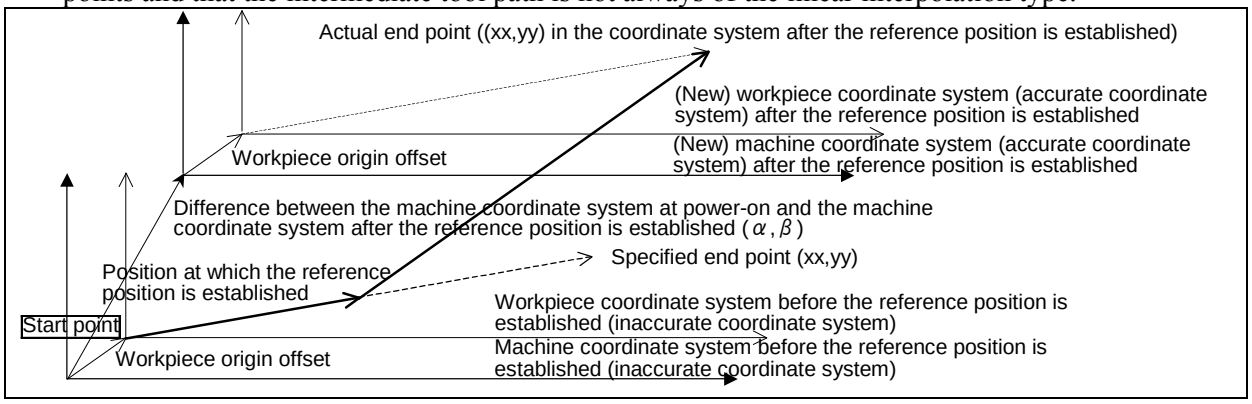


Fig. 4.8.1 (d) Operation when an absolute command is specified

(2) When an incremental command is specified
Movement is made along each axis by a specified distance. (The movement is indicated with the bold line in the figure below. Note that the intermediate tool path is not always of the linear interpolation type.) The coordinates at the end point are those in a newly established coordinate system. When there is a difference (α, β) between the newly established coordinate system and the old coordinate system, the coordinates at the end point are shifted by (α, β). For example, when G91 G00 X100.Y30.; is specified from the position of which absolute coordinates are (0,0), the distance of movement along each axis is (100,30), and the coordinates of the end point are (100-α, 30-β).

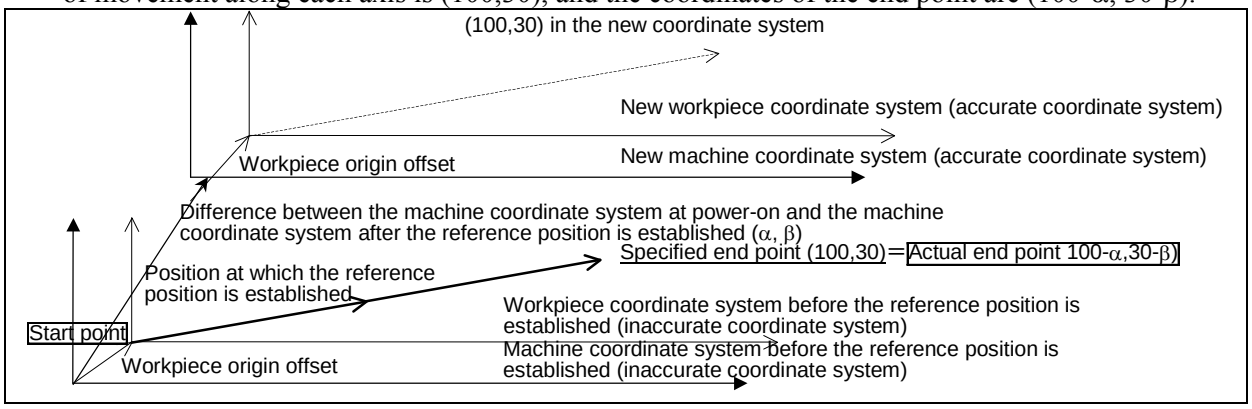


Fig. 4.8.1 (e) Operation when an incremental command is specified

- When a short distance is specified

When a short distance is specified, the end point can be reached before two, three, or four reference marks are passed. Even in such a case, the CNC makes a movement at the FL feedrate while detecting reference marks, but the CNC does not establish the reference position even when the end point is reached.
If the reference position is not established, the next G00 command causes the reference position establishment operation again. The CNC does not use data on the reference marks detected through the previous movement and detects three or four reference marks again to establish the reference position.

- Interruption by feed hold

When a feed hold has been applied during reference position establishment operation, the reference position establishment operation is not performed after execution is restarted. After the restart, non-linear type positioning is performed. In this block, the reference position is not established, so reference position establishment operation is performed again when the next G00 command is specified.

- Interruption by reset or emergency stop

When a reset or emergency stop is applied during reference position establishment operation, the reference position establishment operation is interrupted. Since the reference position is not established, reference position establishment operation is performed again when the next G00 command is specified.

- When an illegal reference mark interval is detected

If a correct reference mark interval cannot be detected for a cause, the tool is positioned to the end point without establishing the reference position. Therefore, the machine position, absolute coordinates, and machine coordinates of the end point are set as if a G00 command without reference position establishment operation were executed. However, the intermediate tool path is different from the tool path obtained by executing the G00 command.
Since the reference position is not established, reference position establishment operation is performed again when the next G00 command is specified.
Even when the CNC detects an illegal reference mark interval, it does not issue the alarm DS1449 "REFERENCE MARK ARE DIFFERENT FROM PARAMETER".

4.8.2 Reference Position Establishment by Jog Feed

Explanation

- Activation condition

If the following conditions are satisfied, reference position establishment operation is performed automatically:

<1> For an axis for which no reference position has been established, feed axis and direction selection signal +Jn or -Jn <G100, G102> is input in jog mode.
<2> The move direction in <1> above matches the reference position return direction set by bit 5 (ZMI) of parameter No. 1006.
<3> The specified axis is not in the following modes:
- Mirror image (mirror image by signal or setting)

NOTE
If all the above conditions are not satisfied, reference position establishment operation is not performed, and the same operation as normal jog feed is performed.

- Operation

The reference position establishment procedure is explained below.

<1> A movement on the specified axis starts at the FL feedrate of reference position return (parameter No.1425).
<2> Upon detection of a reference mark on the scale, the tool is stopped briefly then fed at the reference position return FL feedrate.
<3> Step <2> above is repeated until two, three, or four reference marks are detected on the scale. Then, the reference position is established, and the reference position established establishment signal (ZRF1, ZRF2, ZRF3, etc.) is set to "1".
<4> The tool is fed in the direction selected by the feed axis direction selection signal at the jog feedrate.

When the feed axis and direction selection signal is set to "0" during steps <2> to <4>, feed operation stops. When the feed axis direction selection signal is set to "1" again, the reference position is established.
A time chart for the above procedure is shown below.

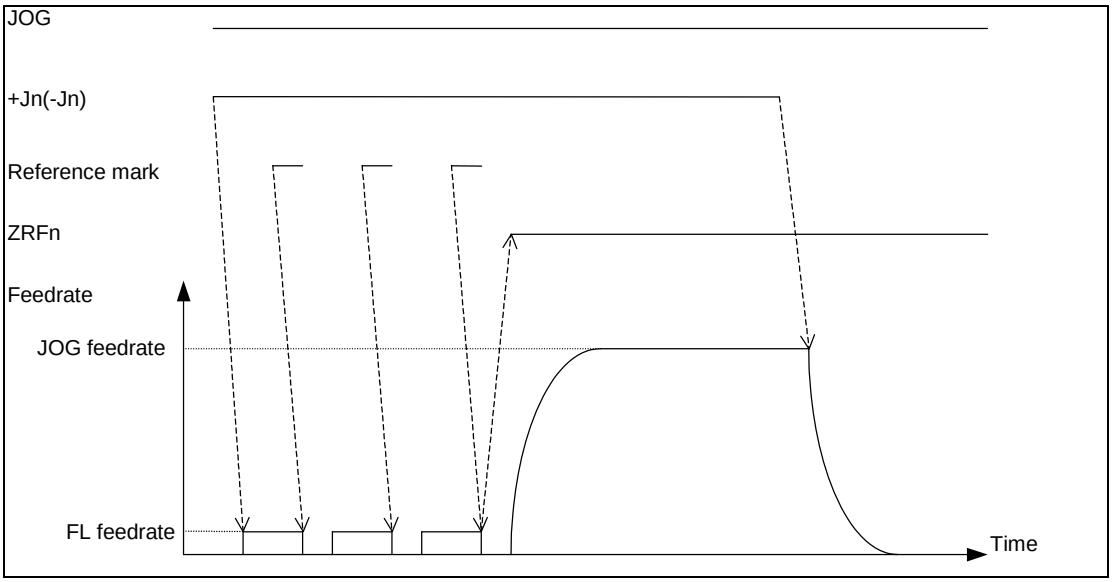


Fig. 4.8.2 (a) Time chart for reference position establishment (jog feed)

- Interruption by the feed axis and direction selection signal

If the feed axis and direction selection signal +Jn (-Jn) is set to "0" during reference position establishment, the reference position establishment operation is interrupted. In this case, data on the reference marks detected through the previous movement is not used for the next reference establishment operation. When the feed axis and direction selection signal +Jn (-Jn) is set to "1" again, the reference position establishment operation is resumed, and the reference position is established after two, three, or four reference marks are detected.

- Interruption by reset or emergency stop

When a reset or emergency stop is applied during reference position establishment operation, axis movement stops, and the reference position establishment operation is interrupted. In this case, data on the reference marks detected through the previous movement is not used for the next reference establishment operation. When the feed axis and direction selection signal +Jn (-Jn) is set to "1" again, the reference position is established after two, three, or four reference marks are detected.

- When an illegal reference mark interval is detected

If a correct reference mark interval cannot be detected for some reason, the tool is positioned at the end point without establishing the reference position. Therefore, the machine position, absolute coordinates, and machine coordinates of the end point are the same as when a G00 command is executed without the reference position establishment operation. However, the intermediate tool path is different from that obtained by executing the G00 command.
Since the reference position is not established, the reference position establishment operation is performed again by the next G00 command.
Even if the CNC detects an illegal reference mark interval, it does not issue the alarm DS1449 "REFERENCE MARK ARE DIFFERENT FROM PARAMETER".

Caution

- PMC axis control

The reference position establishment operation is not performed in rapid traverse (axis control command 00h) or continuous feed (axis control command 06h) under PMC axis control.

- Rapid traverse operation other than G00

In a rapid traverse operation automatically generated by a command such as canned cycle, no reference position establishment operation is carried out.

- Reference position establishment operation

This function has the same specifications as the conventional function of the distance coded linear scale interface, with regard to the operation to detect a reference mark and establish a reference position. The parameter setting, limitation, and the like are also the same.
For details, refer to "Specifications of Linear Scale with Absolute Addressing Reference Marks."

Parameter

	#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						DCLx	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 pulse coder is not used.
1 : A separate pulse coder is used.

NOTE
Set 1 when a distance coded linear scale is used.

#2 DCLx As a separate position detector, the linear scale with a distance coded linear scale
0: Not used.
1: Used.

NOTE
Set 1 when a distance coded linear scale is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1818						DG0x		

[Input type] Parameter input
[Data type] Bit axis

#2 DG0x When the linear scale function with absolute address reference marks is used, reference position establishment operation based on the G00 command and jog feed is:
0: Disabled.
1: Enabled.

4.9 CORRESPONDENCE OF ROTARY SCALE WITHOUT ROTARY DATA

Overview

This manual describes how to deal with an absolute position detector (absolute pulse coder) or a rotary scale with distance-coded reference marks (serial), when the rotary scale without rotary data is used, such as Heidenhain rotary scale RCN723 or Futaba rotary scale FRR902L3DB.
And it is required to set some servo parameters in order to use Heidenhain encoder RCN223, 723 and 220. In the case of using such a scale, please set servo parameters with referring to Item, "Method of using Heidenhain rotary scale RCN223, 723 and 220".

4.9.1 Setting Method by Rotary Axis Type and Movable Range

Rotary axis type	Movable range	Required parameters		Reference item
Rotary axis B type	Under one rotation	No.1817#3	No.1868	Subsec. 4.12.2
Rotary axis B type	Over one rotation	No.1815#0		Subsec. 4.12.3
Rotary axis A type	-	No.1815#6		Subsec. 4.12.4

See the following description for the rotary axial type.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1006							ROSx	ROTx

[Input type] Parameter input
[Data type] Bit axis

ROTx, ROSx Setting linear or rotation 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)

ROSx	ROTx	Meaning
0	1	Rotation axis (A type) (1) Inch/metric conversion is not done. Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by parameter No.1008#0(ROAx) and #2(RRLx). (2) Stored pitch error compensation is the rotation type. (Refer to parameter No.3624) (3) 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	Rotation axis (B type) (1) Inch/metric conversion, absolute coordinate values and relative coordinate values are 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 rotation axis roll over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

4.9.2 In the Case of a Rotary Axis B Type whose Movable Range is under One Rotation

In the case of a rotary axis B type whose movable range is under one rotation with an absolute position detector (absolute pulse coder) or a rotary scale with distance-coded reference marks (serial), there is an uncontinuous point of scale data, if a rotary scale without data (the number of rotation) is used. Then, if an uncontinuous point of scale data is in movable range, the machine coordinate value of NC is different from correct position.
In this function, by the parameters No.1817#3 and No.1868, the machine coordinate value of NC become correct even if an uncontinuous point of scale data is in movable range. Therefore, it is possible to set a rotary axis B type even if an uncontinuous point of scale data is in movable range with a rotary axis B type whose movable range is under one rotation.

Example 1

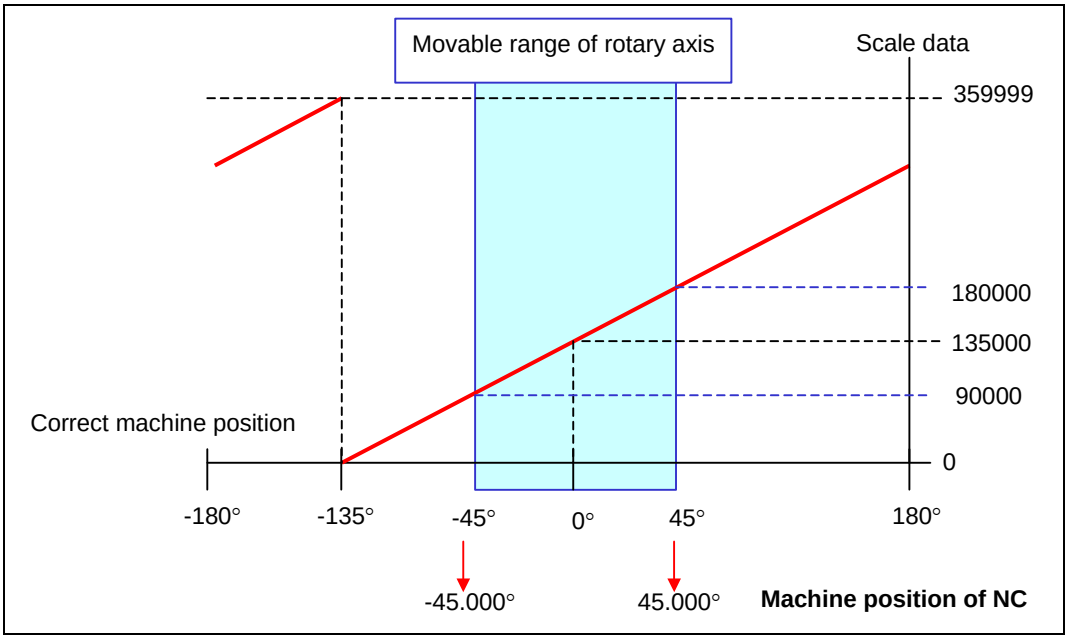


Fig. 4.9.2 (a) The case that the scale data is continuous in movable range of rotary axis

In the case of Fig. 4.9.2 (a), the scale data is continuous from 90000 to 18000.
Because the machine coordinate value of NC is shown by the following equation, (scale data when the power is turned on) - (scale data on the machine coordinate value equals 0), the machine coordinate value of NC becomes correct ones from -45.000 to 45.000.

Example 2

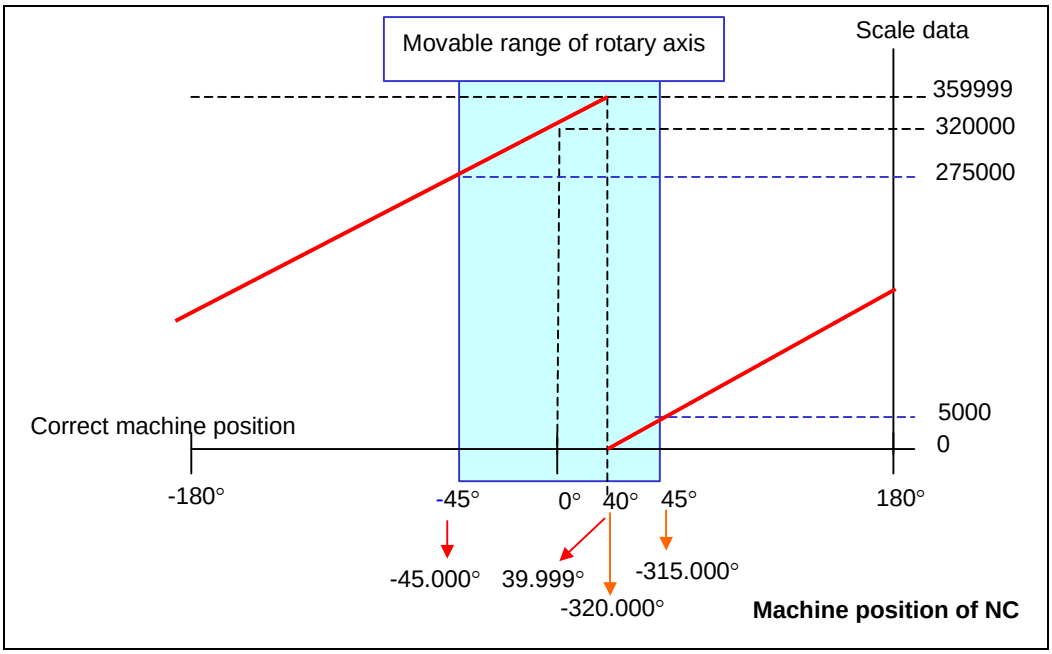


Fig. 4.9.2 (b) The case that the scale data is uncontinuous in movable range of rotary axis.

In the case of Fig. 4.9.2 (b), the scale data of movable range is uncontinuous and separated into two parts, 275000 to 359999 and 0 to 5000, because the uncontinuous point whose machine coordinate value 40.000 exists.

Because the machine coordinate value of NC is shown by the following equation, (scale data when the power is turned on) - (scale data on the machine coordinate value equals 0), NC shows two position ranges. One is correct ones from -45.000 to 0.000 to 39.999. And another is wrong ones from -320.000 to -315.000, which is different from correct position by 360 degree.

Example 3

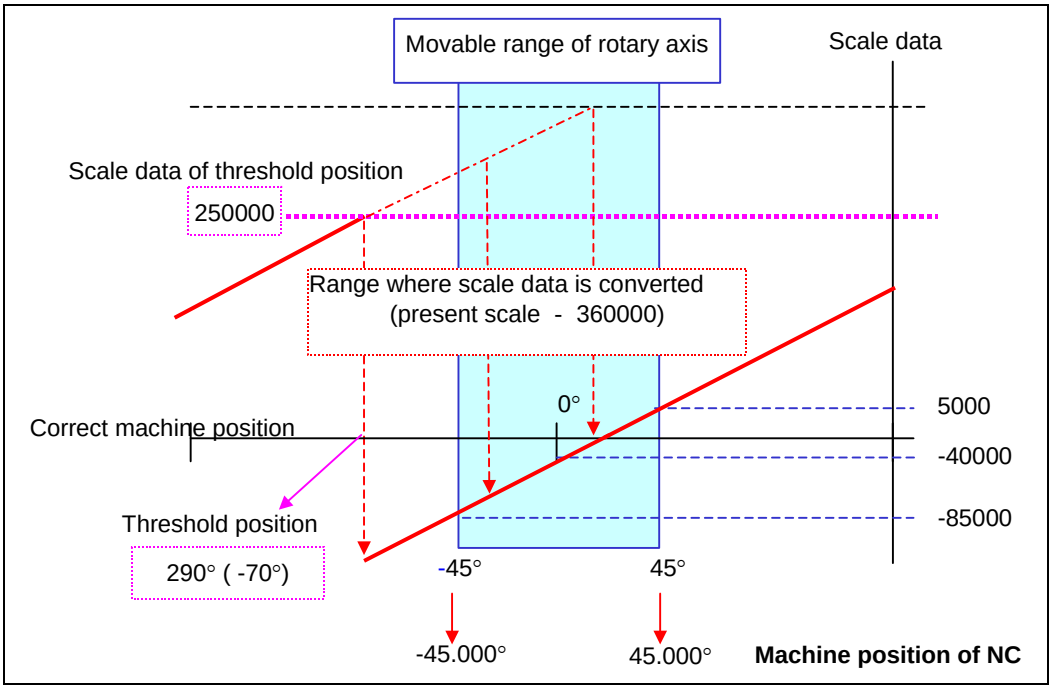


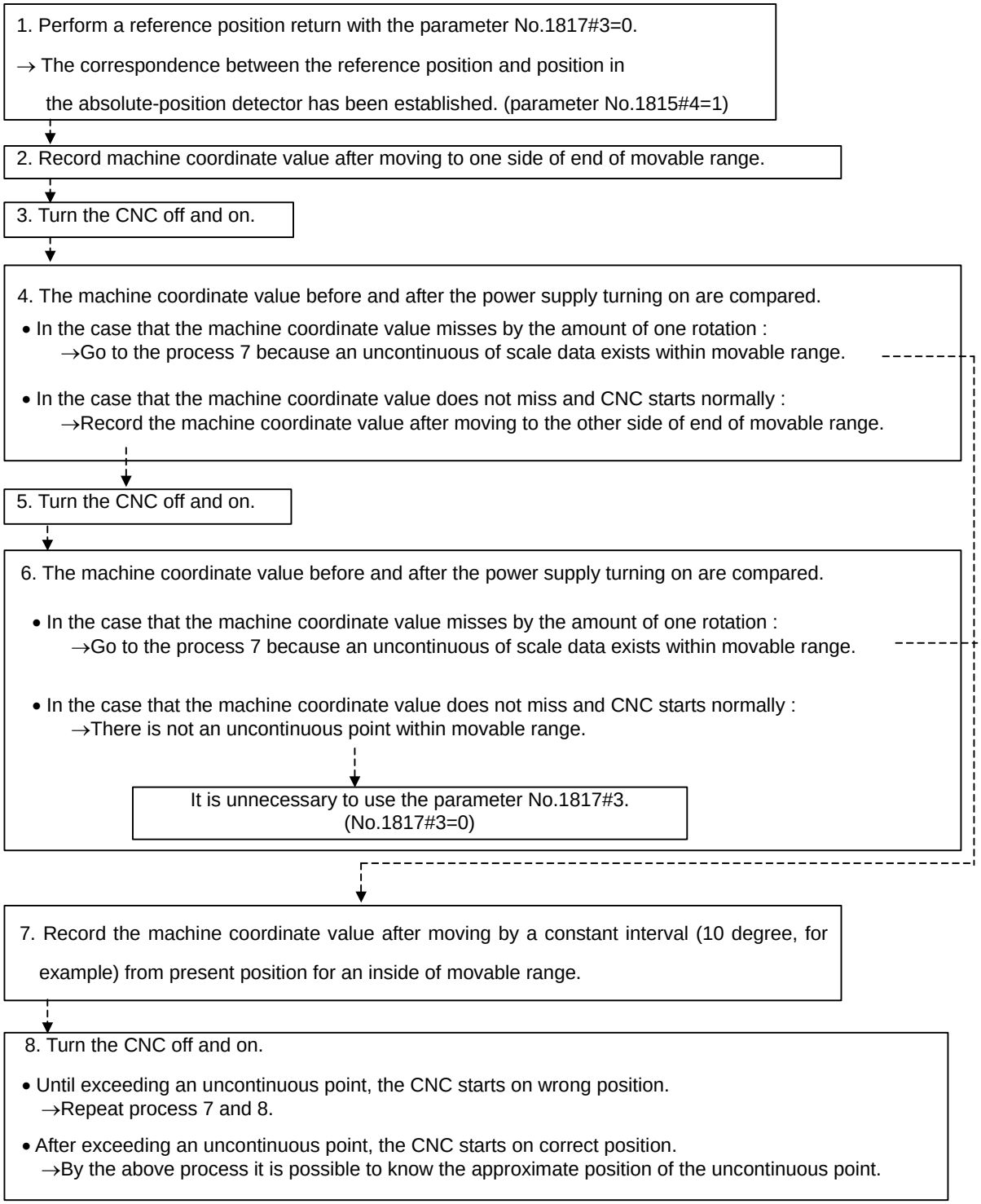
Fig. 4.9.2 (c) The case that parameter No.1817#3 is set to 1 and No.1868 (the threshold position) is set to 290.0.

In the case of Fig. 4.9.2 (b), when 1 is set in bit 3 (SCRx) of parameter No. 1817 and the threshold position (parameter No. 1868) is set to 290.0, the amount of one rotation (360000 in this case) is subtracted from the scale data (250000 to 359999) above the scale data at the threshold position (parameter No. 1868), as shown in Fig. 1.9.2 (c), and the scale data in the movable range of the rotary axis is converted to continuous values -85000 to 5000. Therefore, the scale data is continuous as -85000 to 5000 in movable range and the machine coordinate value of NC can become correct value as -45.000 to 45.000.

The parameter No.1868 (the threshold position) is required to be set the position out of movable range and positive value. (0.0 to 360.0: Set to 290.0 instead of –70.0, in this case.)

An uncontinuous point of scale data

Before using this function, it is necessary to know the position of the uncontinuous point. By the following process, the position of the uncontinuous point can be found in the case of an absolute position detector (absolute pulse coder) (parameter APC(No.1815#5)=1).



In the case of the Fig. 4.9.2 (b) :

1. A reference position return finishing.
2. Moving to one side of movable range (the machine coordinate value -45.000).
3. Turning the CNC off and on.
4. Moving to the other side of movable range (the machine coordinate value is 45.000) because the machine coordinate value at power on is correct.
5. Turning the CNC off and on.
6. Starting on -315.000 with the machine coordinate missing by 360 degree.
7. Shifting by 10 degree and moving to the machine coordinate -325.000.
8. Turning the CNC off and on.

9. Starting correctly on the machine coordinate 35.000.

Thus an uncontinuous point of scale data is found between 35.000 and 45.000.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1817					SCRx			

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#3 SCRx Specifies whether to convert scale data by using threshold position (parameter No.1868) so that rotary axis B type is available, in the case of the axis B type that use a rotary scale without data (the number of rotation), whose movable range is under one rotation:
0 : Not to convert.
1 : To convert.

NOTE
1 This parameter is available for only the rotary axis B type with an absolute position detector (absolute pulse coder) or a rotary scale with distance-coded reference marks (serial).This function cannot be used for distance coded rotary scale interface (phase A/B).
2 This function cannot be used for distance coded rotary scale interface (phase A/B).
3 Don't set this parameter in the case of no uncontinuous point within movable range of rotary axis even if the rotary axis B type.
4 When this parameter is set, machine position and position on absolute position detector become uncorresponding. Consequently, the parameter APZ (No. 1815#4: indicating that the correspondence is established) is set to 0, alarm DS0300. Why the parameter APZ (No. 1815#4) is set to 0 can be checked using diagnostic data No. 310#0.
5 This function cannot be used together with the parameter RVSx (No.1815#0) that save rotary data by CNC, in the case of a rotary axis B type whose movable range is over one rotation.
6 In this function, the amount of one rotation of rotary axis assumes 360, and the machine position 0 assumes the reference position. It is not possible to apply to a rotary axis other than the above-mentioned setting.
7 Set the parameter No.1240 to 0.

1868	Threshold position for converting scale data (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] 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)
In the case that scale data of a rotary scale without rotary data is larger than the scale data of the threshold position (this parameter value), it is converted to be continuous data in movable range by subtracting one rotary data. The position out of movable range (angle from an uncontinuous point) must be set as threshold position. As for the axis with this parameter is set to 0, conversion of scale data is not performed.

NOTE
1 This parameter is available for only the rotary axis B type with an absolute position detector (absolute pulse coder) or a rotary scale with distance-coded reference marks (serial), as for the parameter SCRx(No.1817#3) is set to
2 This function cannot be used for distance coded rotary scale interface (phase A/B).
3 Don't set this parameter in the case of no uncontinuous point within movable range of rotary axis even if the rotary axis B type.
4 When this parameter is set, machine position and position on absolute position detector become uncorresponding. Consequently, the parameter APZ (No. 1815#4: indicating that the correspondence is established) is set to 0, alarm DS0300. Why the parameter APZ (No. 1815#4) is set to 0 can be checked using diagnostic data No. 310#0.

1869	The amount of one rotation of rotary axis B type (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] 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)

Normally, the amount of one rotation of rotary axis is 360, and the machine position 0 is the reference position.
In this case, this parameter is set to 0.

For instance, when this parameter is set to 523.000, the amount of one rotation become 523.000 (in the case of IS-B), if it is necessary to set it arbitrarily.

NOTE
1 When this parameter is set, the power must be turned off before operation is continued.
2 This parameter is available for only the rotary axis B type with an absolute position detector (absolute pulse coder) or a rotary scale with distance-coded reference marks (serial), as for the parameter SCRx(No.1817#3) is set to 1 or the parameter SCRx(No.1815#0) is set to 1.
3 In the case that the amount of one rotation of rotary axis is 360, this parameter is set to 0.If it is necessary to set an amount of one rotation of rotary axis arbitrarily, this parameter is set to the amount of one rotation.
4 When this parameter is set, machine position and position on absolute position detector become uncorresponding. Consequently, the parameter APZ (No. 1815#4: indicating that the correspondence is established) is set to 0, alarm DS0300. Why the parameter APZ (No. 1815#4) is set to 0 can be checked using diagnostic data No. 310#0.
5 This parameter No.1869 is common in movable range that is under one rotation (the parameter SCRx (No.1817#3) is set to 1) and movable range that is over one rotation (the parameter RVS (No.1815#0) is set to 1).

4.9.3 In the Case of a Rotary Axis B Type whose Movable Range is over One Rotation

As for an axis whose movable range is over one rotation and its rotary scale which has no rotary data , for example, 30.000 degree and 390.000 degree have completely same scale data though their position is different from each other by just one rotation. It is possible to set rotary axis B type because CNC saves the rotary data by the following parameter even if in this case.
As attention, a machine coordinate value may get out over a rotation, in the case of moving over 180 degree during turning off, because CNC saves a machine coordinate value just before CNC turns off and in following turning on get from the value.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1815								RVSx

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit axis

#0 RVSx Specifies to save rotary data by CNC, as for an axis whose movable is over one rotation and its rotary scale which has no rotary data:
0: Not to save.
1: To save.

NOTE
1 In the case of a rotary axis B type whose movable range is over one rotation, a rotary scale with rotary data had better be used.
2 When this parameter is set, the power must be turned off before operation is continued.
3 This parameter is available for only the rotary axis B type with an absolute position detector (absolute pulse coder) or a rotary scale with distance-coded reference marks (serial).This function cannot be used for distance coded rotary scale interface (phase A/B).
4 If this parameter is available, the machine coordinate value just before CNC turns off is saved. In the case of moving over 180 degree during turning off, a machine coordinate value may get out over a rotation because CNC saves a machine coordinate value just before CNC turns off and in following turning on get from the value.
5 When this parameter is set, machine position and position on absolute position detector become uncorresponding. Consequently, the parameter APZ (No. 1815#4: indicating that the correspondence is established) is set to 0, alarm DS0300. Why the parameter APZ (No. 1815#4) is set to 0 can be checked using diagnostic data No. 310#0.
6 Absolute coordinate value is set by machine coordinate value. However, after CNC turns on, the workpiece offset such as G92 and G52 executed before CNC turns off is not set.
7 This function cannot be used together with the parameter SCRx (No.1817#3) that convert scale data.
8 In the case that the amount of one rotation of rotary axis is 360, the parameter No.1869 is set to 0. Moreover, set the parameter No.1240 to 0.
9 If it is necessary to set an amount of one rotation of rotary axis arbitrarily, the parameter No.1869 is set to the amount of one rotation. Moreover, set the parameter No.1240 to 0.

1869	The amount of one rotation of rotary axis B type (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] 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)

Normally, the amount of one rotation of rotary axis is 360, and the machine position 0 is the reference position.
In this case, this parameter is set to 0.

For instance, when this parameter is set to 523.000, the amount of one rotation become 523.000 (in the case of IS-B), if it is necessary to set it arbitrarily.

NOTE
1 This parameter is available for only the rotary axis B type with an absolute position detector (absolute pulse coder) or a rotary scale with distance-coded reference marks (serial), as for the parameter SCRx(No.1817#3) is set to 1 or the parameter SCRx(No.1815#0) is set to 1.
2 In the case that the amount of one rotation of rotary axis is 360, this parameter is set to 0.If it is necessary to set an amount of one rotation of rotary axis arbitrarily, this parameter is set to the amount of one rotation.
3 When this parameter is set, machine position and position on absolute position detector become uncorresponding. Consequently, the parameter APZ (No. 1815#4: indicating that the correspondence is established) is set to 0, alarm DS0300. Why the parameter APZ (No. 1815#4) is set to 0 can be checked using diagnostic data No. 310#0.
4 This parameter No.1869 is common in movable range that is under one rotation (the parameter SCRx (No.1817#3) is set to 1) and movable range that is over one rotation (the parameter RVS (No.1815#0) is set to 1).

4.9.4 In the Case of a Rotary Axis A Type

In the case of a rotary axis A type with an absolute position detector (absolute pulse coder), when the machine coordinate passes 0 degree or the shift value per one rotation (360 degree or the parameter No.1260), the data for the reference position (the parameter No.1860, 1861) is renewed so that following up at power on is within one rotation. But as for the absolute position detector (absolute pulse coder) using a scale without rotary data, the data for the reference position (No.1860, 1861) need not be renewed because scale data without rotary data is always read at turning on.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1815		RONx						

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit axis

#6 RONx With a rotation axis A type, an absolute position detector (absolute pulse coder) using a scale without rotary data is:
0: Not used.
1: Used.

NOTE
1 When this parameter is set, the power must be turned off before operation is continued.
2 This parameter is available for only the rotary axis A type with an absolute position detector (absolute pulse coder).This function cannot be used for a rotary scale with distance-coded reference marks (serial) or for a distance coded rotary scale interface (phase A/B).
3 Set it to a rotary axis A type using a scale without rotary data.
4 Do not set it to a rotary axis A type using a scale with rotary data.
5 When this parameter is set, machine position and position on absolute position detector become uncorresponding. Consequently, the parameter APZ (No. 1815#4: indicating that the correspondence is established) is set to 0, alarm DS0300. Why the parameter APZ (No. 1815#4) is set to 0 can be checked using diagnostic data No. 310#0.

4.9.5 Method of Using Heidenhain Rotary Scale RCN223, 723 and 220

Heidenhain rotary scale RCN223, 723 and 220 are the detectors which save absolute position only within one rotation.

- In the case of RCN223, 723 and 220, there is the case that wrong positional data (rotary data) is read on absolute positional communication because it is not saved (it is undefined).

And the former two detectors (RCN223 and 723) are the detectors which have eight million pulses every one rotation.

- In the case of RCN223 and 723, if it is used as usual, eight grids are made every one rotation.

The following parameters are necessary to resolve these problems.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
2275							RCNCLR	800PLS

[Input type] Parameter input
[Data type] Bit axis

#0 800PLS Specifies to use a rotary encorder which has eight million pulses (RCN223, 723 and so on)
0: Not to use
In case of RCN220, this parameter is set to 0.
1: To use (One grid is made every one rotaion)

#1 RCNCLR Specifies to clear rotary data
0: Not to clear
1: To clear (set in using RCN223, 723 and 220)

Set this parameter with the parameter of data mask number (No.2394).

2394	data mask figure number (each axis)

[Input type] Parameter input

[Data type]
[Setting value] 5 or 8
Setting value is according to detector's type. Now a detector on which rotary data must be cleared is RCN223, 723 (set to 8) and RCN220 (set to 5).
Set this parameter with the parameter RCNCLR (No.2275#1).

NOTE
Rotary data of RCN223, 723 and 220 can be saved during separate detector I/F unit turned on. But it is forgotten as soon as it turned off. It is necessary to set parameter in order to clear rotary data because it is varied according to the place where separate detector I/F unit is turned off. And for this reason, RCN223, 723 and 220 can not be used for a linear axis.

- Other parameters that must be set

When you use RCN220, the other settings are the same as normal α/α*i* pulse coders. We show the setting method for RCN223 and RCN723 as follows.

Flexible feed gear

No.2084, 2085
In the case of RCN223, 723, amount of pulses per every one rotation are regarded as eight million pulses.

<flexible feed gear (N/M)> =

$$\frac{\text{Amount of table moving every one rotation of a detector [deg]}}{\frac{\text{Increment of detection [deg]}}{8{,}000{,}000}}$$

Amount of positional pulse

No.2024
amount of positional pulse = 100,000 * (deceleration ratio from a motor to a table)

* As a result of this equation when amount of pulses exceeds input range (0 to 32767), set the positional pulse in following process with using ”positional feed back pulse conversion coefficient”
Amount of positional pulse = A × B
In this case, select B in which A is within 32767. And A, B are set as following.
A : A parameter, amount of positional pulse (within 32767)
No.2024
B : A parameter, positional feed back pulse conversion coefficient
No.2185

Reference counter capacity

No.1821
Set amount of feed back pulses (detection unit) per a table rotation.

* Set amount of (pulses per one table rotation / 8) as reference counter capacity in the case of No.2275#0=0. But in this case eight grids are made every one table rotation.

- Example of parameters setting

System construction

- Heidenhain rotary encorder RCN223
- The least input increment is 1/10,000 degree
- In the case that moving distance is 180 degree every one motor rotation (deceleration ratio is 1/2)

- Deceleration ratio between a rotary table and a separate encorder = 1/1

Setting parameters

- Set No.1815#1=1 so that separate detector is available.
- Set No.2275#0=1, No.2275#1=1, No.2394=8 so that RCN223 is used.
- Parameters for flexible feed gear are calculated.
  Flexible feed gear (N/M) = (360 degree / 0.0001 degree) / 8,000,000 =9 / 20
  So, No.2084=9, No.2085=20
- Amount of positional pulses is calculated.
  Amount of positional pulses = 100,000 × (1 / 2) = 50,000
  Because this value exceeds input range (0 to 32767), following is considered
  50,000 = 12,500 × 4 → A = 12,500 , B=4
  A is set for "amount of positional pulse" and B is set for "positional pulse conversion coefficient"
  No.2024＝12,500 , No.2185=4
- Reference counter capacity is calculated.
  Reference counter capacity = 360 degree / 0.0001 degree = 3,600,000

5 AUTOMATIC OPERATION

Chapter 5, “AUTOMATIC OPERATION”, consists of the following sections:

5.1 CYCLE START/FEED HOLD....381
5.2 RESET AND REWIND....384
5.3 TESTING A PROGRAM....388
5.4 MANUAL ABSOLUTE ON/OFF....393
5.5 OPTIONAL BLOCK SKIP/ADDITION OF OPTIONAL BLOCK SKIP....395
5.6 SEQUENCE NUMBER COMPARISON AND STOP....397
5.7 PROGRAM RESTART....397
5.8 RETRACE (M SERIES)....403
5.9 EXACT STOP / EXACT STOP MODE / TAPPING MODE / CUTTING MODE....413
5.10 RETRACTION FOR RIGID TAPPING....414
5.11 DNC OPERATION....419
5.12 MANUAL INTERVENTION AND RETURN....422
5.13 DIRECT OPERATION BY C LANGUAGE EXECUTOR....425

5.1 CYCLE START/FEED HOLD

Overview

- Start of automatic operation (cycle start)

When automatic operation start signal ST is set to 1 then 0 while the CNC is in memory (MEM) mode, DNC operation mode (RMT), or manual data input (MDI) mode, the CNC enters the automatic operation start state then starts operating.

Signal ST, however, is ignored in the following cases:

1. When the mode is other than MEM, RMT, or MDI
2. When the feed hold signal (*SP) is set to "0"
3. When the emergency stop signal (*ESP) is set to "0"
4. When the external reset signal (ERS) is set to "1"
5. When the reset and rewind signal (RRW) is set to "1"
6. When MDI RESET key is pressed
7. When the CNC is in the alarm state
8. When the CNC is in the NOT READY state
9. When automatic operation is executing
10. When the program restart signal (SRN) is "1"
11. When the CNC is searching for a sequence number.

The CNC enters the feed hold state and stops operation in the following cases during automatic operation:

1. When the feed hold signal (*SP) is set to "0"
2. When the mode is changed to manual operation mode (JOG, INC, HND, REF, TJOG, or THND).

The CNC enters the automatic operation stop state and stops operating in the following cases during automatic operation:

1. When a single command block is completed during a single block operation
2. When operation in manual data input (MDI) mode has been completed
3. When an alarm occurs in the CNC
4. When a single command block is completed after the mode is changed to other automatic operation mode or memory edit (EDIT)

The CNC enters the reset state and stops operating in the following cases during automatic operation:

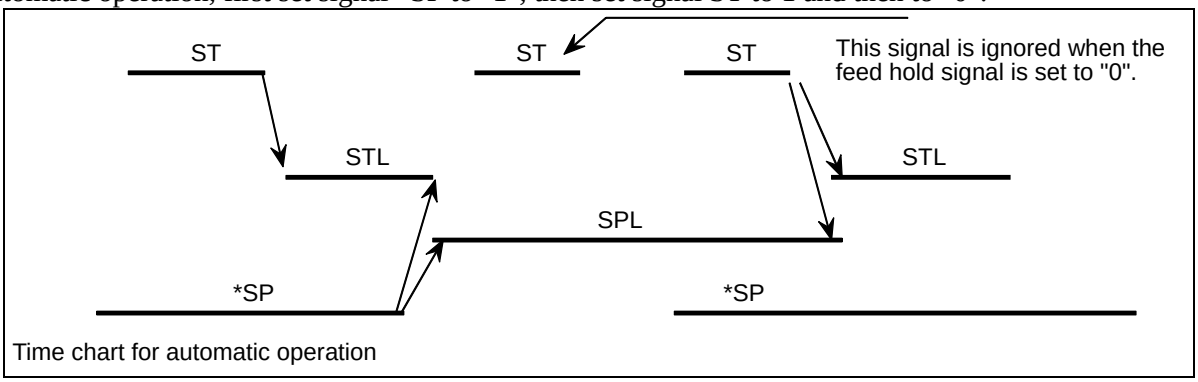
1. When the emergency stop signal (*ESP) is set to "0"
2. When the external reset signal (ERS) is set to "1"
3. When the reset and rewind signal (RRW) is set to "1"

4. When MDI RESET key is pressed

The state of the CNC (automatic operation start, feed hold, automatic operation stop, or reset) is posted to the PMC with status output signals OP, SPL, and STL. See the table in the "Signal" section for details.

- Halt of automatic operation (feed hold)

When the feed hold signal *SP is set to 0 during automatic operation, the CNC enters the feed hold state and stops operation. At the same time, cycle start lamp signal STL is set to "0" and feed hold lamp signal SPL is set to "1". Re-setting signal *SP to "1" in itself will not restart automatic operation. To restart automatic operation, first set signal *SP to "1", then set signal ST to 1 and then to "0".



Time chart for automatic operation

When signal *SP is set to "0" during the execution of a block containing only the M, S, T, or B function, signal STL is immediately set to "0", signal SPL is set to "1", and the CNC enters the feed hold state. If the FIN signal is subsequently sent from the PMC, the CNC executes processing up until the end of the block that has been halted. Upon the completion of that block, signal SPL is set to "0" (signal STL remains set to "0") and the CNC enters the automatic operation stop state.

(a) During threading
When signal *SP is set to "0" during threading, the CNC enters the feed hold state after executing a non-threading block after the threading blocks.
In lathe system, when signal *SP is set to "0" during threading with the G92 command (threading cycle), signal SPL is immediately set to "1" but operation continues up until the end of the retraction block following threading. When signal *SP is set to "0" during threading with the G32 (M series: G33) command, signal SPL is immediately set to "1" but operation continues until the end of a non-threading block following the threading blocks. (Stopping feeding during threading is dangerous because the amount of cutting will increase.)

(b) During tapping in a canned cycle (G84)
When signal *SP is set to "0" during tapping in a canned cycle (G84), signal SPL is immediately set to "1" but operation continues until the tool returns to the initial level or R point level after the completion of tapping.

(c) When a macro instruction is being executed
Operation stops after the currently executing macro instruction has been completed.

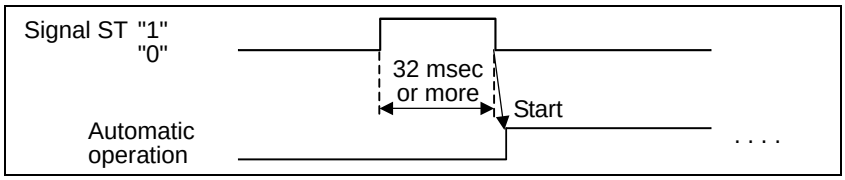
Signal

Cycle start signal ST<Gn007.2>

[Classification] Input signal

[Function] Starts automatic operation.

[Operation] When signal ST is set to "1" then "0" in memory (MEM) mode, DNC operation mode (RMT) or manual data input (MDI) mode, the CNC enters the cycle start state and starts operation.
MEM, RMT, or MDI mode



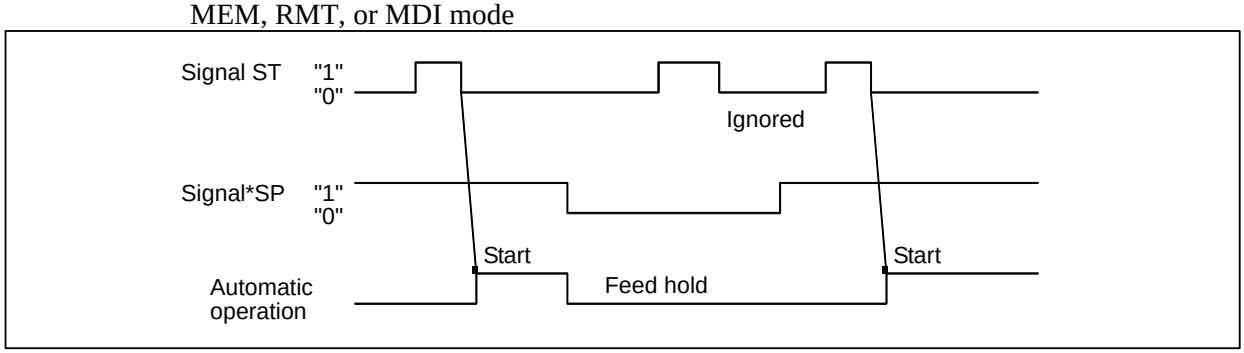
Feed hold signal *SP<Gn008.5>

[Classification] Input signal

[Function] Halts automatic operation.

[Operation] When signal *SP is set to "0" during automatic operation, the CNC enters the feed hold state and stops operation. Automatic operation cannot be started when signal *SP is set to "0".

MEM, RMT, or MDI mode



Automatic operation signal OP<Fn000.7>

[Classification] Output signal

[Function] Notifies the PMC that automatic operation is in progress.

[Output cond.] This signal is set to "1" or "0", according to the state of the CNC, as listed in Table 5.1 (a).

Cycle start lamp signal STL<Fn000.5>

[Classification] Output signal

[Function] Notifies the PMC that automatic operation start is entered.

[Output cond.] This signal is set to "1" or "0", according to the state of the CNC, as listed in Table 5.1 (a).

Feed hold lamp signal SPL<Fn000.4>

[Classification] Output signal

[Function] Notifies the PMC that feed hold state is entered.

[Output cond.] This signal is set to "1" or "0", according to the state of the CNC, as listed in Table 5.1 (a).

Signals OP, STL, and SPL are the signals to inform PMC of the operation condition of CNC.

Table 5.1 (a) Status of operation

Signal name / State of the operation	Cycle start lamp STL	Feed hold lamp SPL	Automatic operation lamp OP
Cycle start state	1	0	1
Feed hold state	0	1	1
Automatic operation stop state	0	0	1
Reset state	0	0	0

(a) Cycle start state
The CNC is executing memory operation or manual data input operation commands.
(b) Feed hold state
The CNC is not executing memory operation nor manual data input operation commands while the commands to be executed remain.
(c) Automatic operation stop state
Memory operation or manual data input operation has been completed and stopped.
(d) Reset state
The automatic operation has been forcibly terminated.

NOTE
If bit 2 (OPS) of parameter No. 11223 is set to 1, signal OP is set to "1" when a sequence number is searched for by operating keys on the MDI panel in the automatic operation mode (MEM).

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn007						ST		
Gn008			*SP					

	#7	#6	#5	#4	#3	#2	#1	#0
Fn000	OP		STL	SPL				

Alarm and message

- Self-diagnosis information

During automatic operation, the machine may sometimes show no movement while no alarm is detected. In that case, the CNC may be performing processing or waiting for the occurrence of an event. The state of the CNC can be obtained using the CNC self-diagnosis function (diagnosis numbers 1000 to 1015).

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Memory operation
	MDI operation
	DNC operation
	Checking by diagnostic display

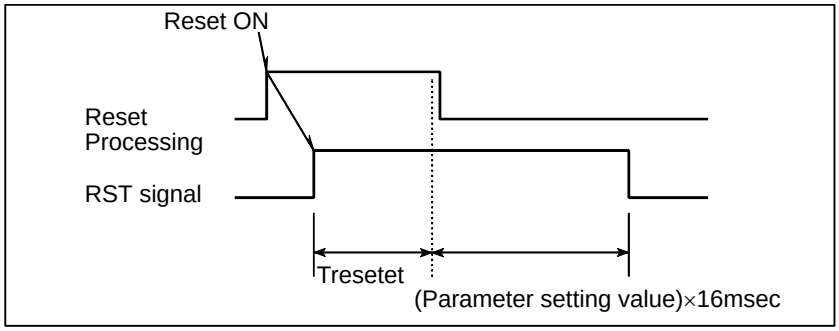
5.2 RESET AND REWIND

Overview

The CNC is reset and enters the reset state in the following cases:
1. When the emergency stop signal (*ESP) is set to "0"
2. When the external reset signal (ERS) is set to "1"
3. When the reset and rewind signal (RRW) is set to "1"
4. When MDI RESET key is pressed

When the CNC is reset, the resetting signal (RST) is output to the PMC. The resetting signal (RST) is set to "0" when the resetting signal output time, set with parameter No. 3017, has elapsed after the above conditions have been released.

RST signal output time = Treset (Reset processing time) + (parameter setting value) × 16 msec.



⚠ CAUTION
Treset requires at least 16 msec. This time will be longer on optional configurations.

When the CNC is reset during automatic operation, automatic operation is stopped and tool movement along the controlled axis is decelerated and stopped(*1). When the CNC is reset during the execution of the M, S, T, or B function, signal MF, SF, TF, or BF is set to "0" within 100 ms.
Tool movement along the controlled axis is also decelerated and stopped(*1) in manual operation (jog feed, manual handle feed, incremental feed, or etc).

⚠ CAUTION
*1 When the emergency stop signal (*ESP) is set to "0", the tool is stopped by an emergency stop.

Parameter CLR (No. 3402#6) is used to select whether the CNC internal data (such as modal G codes) is cleared or reset when the CNC is reset. Refer to the Appendix E, "Status when turning on power, when cleared, and when reset" in the Operator's Manual for the state of the internal data when cleared or reset.
The following parameters are also used to select how to handle processing for CNC data when the CNC is reset.

- Bit 7 (MCL) of parameter No. 3203
  A program prepared in the MDI mode is:
  0: Not cleared
  1: Cleared
- Bit 6 (CCV) of parameter No. 6001
  Custom macro variables #100 to #199 are:
  0: Cleared
  1: Not cleared

- Reset & Rewind

When the reset & rewind signal RRW becomes "1", a reset and rewinding are performed.
Rewinding is described below.

1. When a portable tape reader is connected to the selected input/output equipment in DNC operation mode, the portable tape reader is rewound.
   The rewinding signal RWD is output while the portable tape reader is being rewound. This signal becomes "0" when rewinding is completed.
2. In cases other than the above, the beginning of the selected main program is found. RWM (bit 2 of parameter No. 3001) can be used to specify whether the rewinding signal RWD is output.
   When RWM (bit 2 of parameter No. 3001) is 1, the rewinding signal RWD is kept "1" for 100 ms and then becomes "0". Since the time required for finding the beginning of the main program in memory is almost zero, the beginning is already found by the time the rewinding signal RWD becomes "0".

Signal

External reset signal ERS<Gn008.7>

[Classification] Input signal
[Function] Reset the CNC.
[Operation] Turning the signal ERS to "1" resets the CNC and enters the reset state. While the CNC is reset, the resetting signal RST turns to "1".

Reset & rewind signal RRW<Gn008.6>

[Classification] Input signal
[Function] CNC is reset and a program under an automatic operation is rewound.
[Operation] As described in the item, "Reset & Rewind".

Resetting signal RST<Fn001.1>

[Classification] Output signal
[Function] Notifies the PMC that the CNC is being reset. This signal is used for reset processing on the PMC.
[Output cond.] This signal is set to "1" in the following cases:
1. When the emergency stop signal (*ESP) is set to "0"
2. When the external reset signal (ERS) is set to "1"
3. When the reset & rewind signal (RRW) is set to "1"
4. When MDI RESET key is pressed

This signal is set to "0" in the following case:
When the resetting signal output time, set with parameter No. 3017, has elapsed after the above conditions have been released and the CNC is reset

Rewinding signal RWD<Fn000.0>

[Classification] Output signal
[Function] Notifies the PMC that the CNC is being rewound.
[Output cond.] As described in the item, "Reset and Rewind".

MDI reset confirmation signal MDIRST<Fn006.1>

[Classification] Output signal
[Function] Notifies the PMC that the RESET MDI key has been pressed.
[Output cond.] This signal is set to "1" when:
- The RESET key is pressed when a path for which MDI reset is enabled is selected.

This signal is set to "0" when:
- The RESET MDI key is not pressed.
- A path for which MDI reset is not enabled is selected.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn008	ERS	RRW						

	#7	#6	#5	#4	#3	#2	#1	#0
Fn000								RWD
Fn001							RST	
Fn006							MDIRST	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3001						RWM		

[Input type] Parameter input
[Data type] Bit path

#2 **RWM** While a program in the program memory is being searched for, the rewind signal (RWD) is:
0: Not output.
1: Output.

3017	Output time of reset signal RST

[Input type] Parameter input
[Data type] Word path
[Unit of data] 16msec
[Valid data range] 0 to 255
When the output time of the reset signal RST is to be extended, set an extended time.
(RST signal output time) =
(Time required for reset processing) + (Parameter setting) × 16 msec

	#7	#6	#5	#4	#3	#2	#1	#0
3203	MCL							

[Input type] Parameter input
[Data type] Bit path

#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
3402		CLR						

[Input type] Parameter input
[Data type] Bit path

#6 CLR Reset button on the MDI panel, 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	#6	#5	#4	#3	#2	#1	#0
6001		CCV						

[Input type] Parameter input
[Data type] Bit path

#6 CCV Common variables #100 to #199 cleared by power-off are:
0: Cleared to <null> by reset
1: Not cleared by reset

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Status when turning power on, when clear and when reset

5.3 TESTING A PROGRAM

Overview

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 or viewing the position display change without running the machine.

5.3.1 Machine Lock

Overview

The change of the position display can be monitored without moving the machine. When all-axis machine lock signal MLK, or each-axis machine lock signals MLK1 to MLK5 are set to "1", output pulses (move commands) to the servo motors are stopped in manual or automatic operation. The commands are distributed, however, updating the absolute and relative coordinates. The operator can therefore check if the commands are correct by monitoring the position display.

Signal

All-axis machine lock signal MLK<Gn044.1>

[Classification] Input signal

[Function] Places all controlled axes in the machine lock state.

[Operation] When this signal is set to "1", pulses (move commands) are not output to the servo motors for all axes in manual or automatic operation.

Each-axis machine lock signals MLK1 to MLK5 <Gn108.0 to Gn108.4>

[Classification] Input signal

[Function] Place the corresponding controlled axes in the machine lock state.
These signals are provided for each controlled axis. The signal number corresponds to the number of the controlled axis.

MLK 1

1 . . . Machine lock for the 1st axis
2 . . . Machine lock for the 2nd axis
3 . . . Machine lock for the 3rd axis
:

[Operation] When these signals are set to "1", pulses (move commands) are not output to the servo motors for the corresponding axes (1st to 5th) in manual or automatic operation.

All-axis machine lock check signal MMLK<Fn004.1>

[Classification] Output signal

[Function] Notifies the PMC of the state of the all-axis machine lock signal.

[Output cond.] This signal is set to "1" in the following case:
- When all-axis machine lock signal MLK is set to "1"

This signal is set to "0" in the following case:
- When all-axis machine lock signal MLK is set to "0"

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn044							MLK	
Gn108				MLK5	MLK4	MLK3	MLK2	MLK1
	#7	#6	#5	#4	#3	#2	#1	#0
Fn004							M MLK	

Note

NOTE

1 Automatic operation in the machine lock state (M, S, T, and B commands)
Machine lock applies only to move commands along controlled axes. Updating modal G codes or setting a coordinate system is performed normally. M, S, T, and B (2nd auxiliary function) commands are also performed normally.

2 Reference position return in the machine lock state (G27, G28, and G30)
When the reference position return command (G28), or 2nd to 4th reference position return command (G30), is executed for an axis in the machine lock state, distribution and position updating are performed. The tool, however, is not returned to the reference position. The reference position return completion signals (ZP1 to ZP5) are not output.
The reference position return check command (G27) is ignored in the machine lock state.

3 Turning on/off the machine lock signal during movement along an axis
When the machine lock signal for an axis is set to 1 during movement along the axis that is not in the machine lock state, the axis is immediately placed in the machine lock state and output pulses (move commands) to the servo motor are stopped. The tool is decelerated and stopped with the automatic acceleration/deceleration function.
On the other hand, when the machine lock signal for an axis is set to "0" during distribution of the move command along the axis in the machine lock state, pulse (move command) output for the axis is immediately restarted. The tool is accelerated with the automatic acceleration/deceleration function.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Machine lock and auxiliary function lock

5.3.2 Dry Run

Overview

Dry run is valid only for automatic operation.
The tool is moved at a constant feedrate(*1) regardless of the feedrate specified in the program. This function is used, for example, to check the movement of the tool without a workpiece.

⚠ CAUTION

This feedrate depends on the specified parameters, the manual rapid traverse switching signal (RT), manual feedrate override signals (*JV0 to *JV15), and whether the command block specifies rapid traverse or cutting feed, as listed in the table below.

Manual rapid traverse switching signal (RT)	Program command	
	Rapid traverse	Cutting feed
1	Rapid traverse rate	Dry run feedrate × Jvmax (*2)
0	Dry run speed × 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
JV : Manual feedrate override
Jvmax : Maximum value of manual feedrate override

*1 Dry run feedrate # JV when parameter RDR (No. 1401#6) is 1. Rapid traverse rate when parameter RDR is 0.
*2 Clamped by maximum cutting feedrate.

Signal

Dry run signal DRN<Gn046.7>

[Classification] Input signal
[Function] Enables dry run.
[Operation] When this signal is set to "1", the tool is moved at the feedrate specified for dry run.
When this signal is set to "0", the tool is moved normally.

> ⚠ **CAUTION**
> When the dry run signal is changed from "0" to "1" or "1" to "0" during the movement of the tool, the feedrate of the tool is first decelerated to "0" before being accelerated to the specified feedrate.

Dry run check signal MDRN<Fn002.7>

[Classification] Output signal
[Function] Notifies the PMC of the state of the dry run signal.
[Output cond.] This signal is set to "1" in the following case:
- When dry run signal DRN is set to "1"

This signal is set to "0" in the following case:
- When dry run signal DRN is set to "0"

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn046	DRN							
Fn002	MDRN							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401		RDR	TDR					

[Input type] Parameter input
[Data type] Bit path

#5 TDR Dry run during threading or tapping (tapping cycle G74 or G84, rigid tapping)
0: Enabled
1: Disabled

#6 RDR Dry run for rapid traverse command
0: Disabled
1: Enabled

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.

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.

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.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Dry run

5.3.3 Single Block

Overview

Single block operation is valid only for automatic operation.
When the single block signal (SBK) is set to "1" during automatic operation, the CNC enters the automatic operation stop state after executing the current block. In subsequent automatic operation, the CNC enters the automatic operation stop state after executing each block in the program. When the single block signal (SBK) is set to "0", normal automatic operation is restored.
Single block operation during the execution of custom macro statements depends on the setting of parameter SBM (No. 6000#5) or SBV (No. 6000#7), as follows:
SBM=0, SBV=0:

Operation does not stop in the custom macro statements but stops once the next NC command has been executed.

SBM=1:
Operation stops after each block in the custom macro statements.
SBV=1:
Single block operation in custom macro statements is suppressed using macro system variable #3003. Execution of custom macro statements is stopped after block execution.
When the CNC is in the automatic operation stop state during single block operation, the mode can be changed to manual data input (MDI), manual handle feed (HNDL), incremental feed (INC), or jog feed (JOG), by using the mode select signals (MD1, MD2, and MD4).

Signal

Single block signal SBK<Gn046.1>

[Classification] Input signal
[Function] Enables single block operation.
[Operation] When this signal is set to "1", single block operation is performed. When this signal is set to "0", normal operation is performed.

Single block check signal MSBK<Fn004.3>

[Classification] Output signal
[Function] Notifies the PMC of the state of the single block signal.
[Output cond.] This signal is set to "1" in the following case:
- When single block signal SBK is set to "1"

This signal is set to "0" in the following case:
- When single block signal SBK is set to "0"

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn046							SBK	
Fn004					MSBK			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6000	SBV		SBM					

[Input type] Parameter input
[Data type] Bit path

#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 SBV Custom macro statement
0: Not stop the single block
1: Enable/disable single block stop with system variable #3003

		Parameter SBM (No.6000#5)	
		0	1
Parameter SBV (No.6000#7)	0	Disables single block stop.	Enables single block stop. (With variable #3003, single block stop cannot be enabled/disabled. Single block stop is enabled at all times.)
	1	Enables single block stop. (With variable #3003, single block stop can be enabled/disabled.)	

CAUTION

⚠ CAUTION

1 Operation in thread cutting
When the SBK signal turns to "1" during thread cutting, operation stops after execution of the first non-thread cutting block after the thread cutting command.

2 Operation in canned cycle
When the SBK signal turns to "1" during canned cycle operation, the operation stops at each positioning, approach, drilling and retreat instead of the end of the block. The SPL signal turns to "1" while the STL signal turns to "0", showing that the end of the block has not been reached. When the execution of one block is completed, the STL and SPL signals turn to "0" and the operation is stopped.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Single block

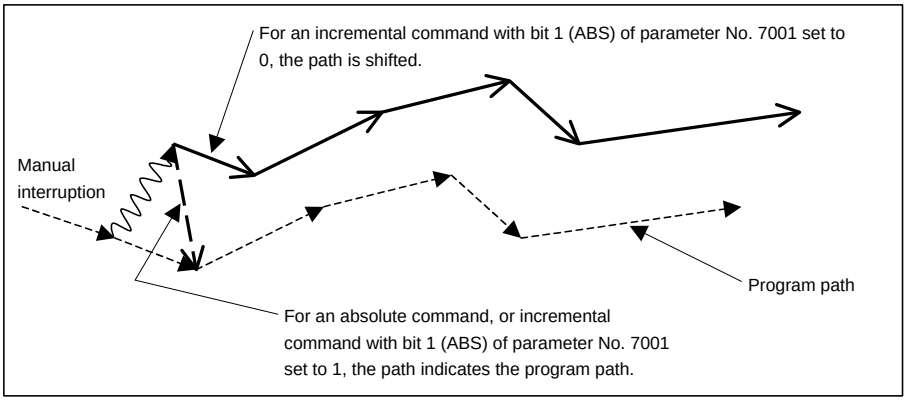
5.4 MANUAL ABSOLUTE ON/OFF

Overview

Whether the distance by which the tool is moved by manual operation (such as jog feed or manual handle feed) is reflected in the absolute coordinate system can be selected.
Also, a confirmation signal indicating the CNC manual absolute on/off state is output.

- When manual absolute turns on (Manual absolute signal *ABSM=0)

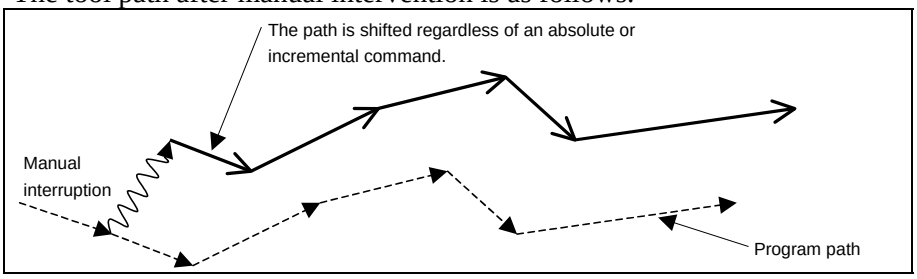
When manual intervention is performed during automatic operation, the distance by which the tool is moved by manual intervention is reflected in the absolute coordinate system. For this reason, the absolute and machine coordinate systems are the same before and after manual intervention.
The tool path after manual intervention is determined according to the setting of bit 1 (ABS) of parameter No. 7001 as follows.



- When manual absolute turns off (Manual absolute signal *ABSM=1)

The distance by which the tool is moved is reflected in the absolute position display during manual operation.

When the system is reset or restarted in the automatic operation mode after manual operation, however, the absolute position display indicates the position before manual intervention and the absolute coordinate system after manual intervention is shifted from the machine coordinate system by the amount of manual intervention. The tool path after manual intervention is as follows.



Signal

Manual absolute signal *ABSM<Gn006.2>

[Classification] Input signal

[Function] Turns the manual absolute function on or off.

[Operation] When this signal is set to 0, turns on the manual absolute function.
When this signal is set to 1, turns off the manual absolute function.

Manual absolute check signal MABSM<Fn004.2>

[Classification] Output signal

[Function] Notifies the state of the manual absolute signal.

[Output cond.] This signal is set to 1 in the following case:
- When the manual absolute signal *ABSM is set to 0

This signal is set to 0 in the following case:
- When manual absolute signal *ABSM is set to 1

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn006						*ABSM		
Fn004						MABSM		

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7001							ABS	

[Input type] Parameter input

[Data type] Bit path

#1 ABS The move command issued after manual intervention when manual absolute is on uses:
0: Different paths between absolute (G90) and incremental (G91).
1: Same path (path for absolute) between absolute (G90) and incremental (G91).

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Manual absolute on/off

5.5 OPTIONAL BLOCK SKIP/ADDITION OF OPTIONAL BLOCK SKIP

Overview

When a slash followed by a number (/n, where 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 contained in the block for which /n, corresponding to signal BDTn, is specified is ignored (from /n to the end of the block).

(Example) /2 N123 X100.0 Y200.0 ;

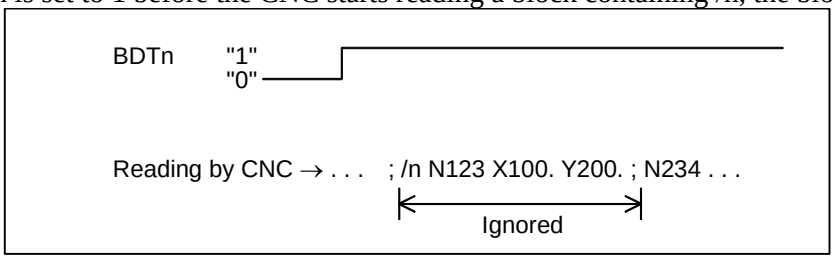
Input signal	Code specified at the head of a block
BDT1	/ or /1 (Note)
BDT2	/2
BDT3	/3
BDT4	/4
BDT5	/5
BDT6	/6
BDT7	/7
BDT8	/8
BDT9	/9

NOTE
Number 1 for /1 can be omitted. However, when two or more optional block skip switches are used in one block, number 1 for /1 cannot be omitted.

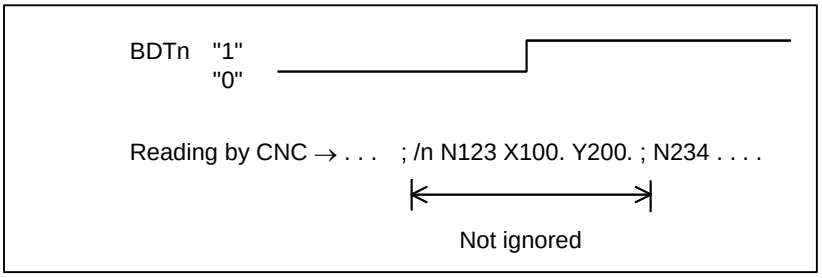
(Example)
//3 N123 X100.0 Y200.0 ; Invalid
/1 /3 N123 X100.0 Y200.0 ; Valid

The following figures show the relationship between the timing, when optional block skip signals (BDT1 to BDT9) are set to 1, and the ignored information:

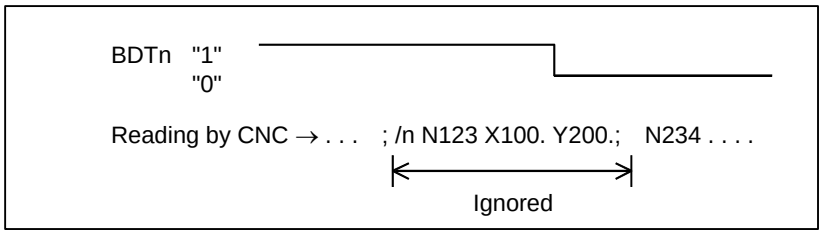
1. When BDTn is set to 1 before the CNC starts reading a block containing /n, the block is ignored.



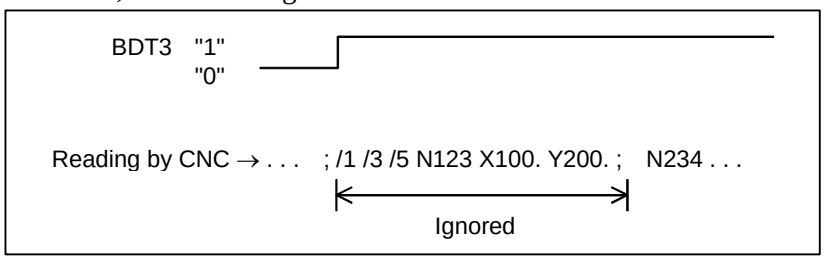
2. When BDTn is set to 1 while the CNC is reading a block containing /n, the block is not ignored.



3. When BDTn, currently set to 1, is set to 0 while the CNC is reading a block containing /n, the block is ignored.



4. When two or more optional block skip switches are specified in a block and BDTn, corresponding to one of them, is set to 1, the block is ignored.



Signal

Optional block skip signals BDT1<Gn044.0>, BDT2 to BDT9<Gn045>

[Classification] Input signal

[Function] Select whether a block containing /n is to be executed or ignored.

[Operation] During automatic operation, a block containing /n in the program is ignored when the corresponding optional block skip signal is set to 1. It is executed normally when the signal is set to 0.

Optional block skip check signals MBDT1<Fn004.0>, MBDT2 to MBDT9<Fn005>

[Classification] Output signal

[Function] Notify the PMC of the states of the optional block skip signals BDT1 to BDT9. Nine signals are provided, corresponding to the nine optional block skip signals. Signal MBDTn corresponds to signal BDTn.

[Output cond.] Signal MBDTn is set to 1 in the following case:

- When the corresponding optional block skip signal (BDTn) is set to 1

Signal MBDTn is set to 0 in the following case:

- When the corresponding optional block skip signal (BDTn) is set to 0

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn044								BDT1
Gn045	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2

	#7	#6	#5	#4	#3	#2	#1	#0
Fn004								MBDT1
Fn005	MBDT9	MBDT8	MBDT7	MBDT6	MBDT5	MBDT4	MBDT3	MBDT2

Note

NOTE

1 This function is ignored when programs are loaded into memory. Blocks containing /n are also stored in memory, regardless of how the optional block skip signal is set.
Programs stored in memory can be output, regardless of how the optional block skip signals are set.
Optional block skip is effective even during sequence number search operation.

2 Position of a slash
A slash (/) must be specified at the head of a block. 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 an optional block skip signal is "1". TH and TV checks are made for the skipped portions in the same way as when the optional block skip switch is "0".

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Program section configuration

5.6 SEQUENCE NUMBER COMPARISON AND STOP

Overview

When a block having the preset sequence number is found in a running program, the tool enters the single block stop state after executing that block.

Explanation

In the MDI mode, set the number of the program (1 to 9999) containing the sequence number at which to stop the program and the sequence number at which to stop the program (1 to 99999) in "Comparison and Stop (Program)" and "Comparison and Stop (Sequence)" in the setting data, respectively.
During the automatic operation, single block stop occurs after the block containing the preset sequence number is executed.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Sequence number comparison and stop

5.7 PROGRAM RESTART

Overview

When the tool is broken during automatic operation, or when a machining operation interrupted by a holiday is to be restarted after the holiday, you can restart machining from a desired block by specifying the sequence number of the block or the number of blocks from the beginning of the program to the block where machining is to restart.
This function can also be used as a high-speed program check function.
P type: Restart after a tool is broken down
Q type: Restart after the power is turned off (after a holiday, etc.) or after emergency stop is canceled

NOTE
To use the program restart function, set 0 in bit 2 (NSQ) of parameter No. 8135.

Explanation

- Outputting M, S, T, and B (second auxiliary function) codes on the program restart screen

By setting bit 7 (MOP) of parameter No. 7300, the M, S, T, and B (second auxiliary function) codes can be output after a block where the program is to restart is searched for.

1 Before a movement to the machining restart point

<1> The last M, S, T, and B (2nd auxiliary function) codes can be output to the PMC automatically. The last S code is output as the maximum spindle speed if the S code is specified together with G92 in one block; otherwise, the S code is output as a specified spindle speed. On the program restart screen, only the S code specified last is indicated regardless of whether it is specified together with G92 in one block or not.

<2> All M codes sampled during the search for the block where the program is to restart and the last S, T, and B (second auxiliary function) codes can be output to the PMC automatically. Up to 35 M cods can be sampled. If there are more than 35 M codes to be sampled, the latest 35 M codes are output to the PMC.

<1> or <2> above can be selected by setting bit 6 (MOA) of parameter No. 7300.

2 Until the machining restart point is reached (over store mode)

On the program restart screen, you can specify M, S, T, and B (second auxiliary function) codes through MDI key input while maintaining the MEM mode or DNC mode.

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 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 signal

When program restart signal SRN <Gn006.0> is set to "1", pressing the cycle start does not cause a start.

- 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 DS024 is issued.

- Commands that prevent program restart

Program restart cannot be performed for blocks placed in the following modes:

- Polar coordinate interpolation (G12.1) (T series)
- Threading (G32,G33), Threadingcycle (G92), Multiple repetitive threading cycle (G76) (T series)
- Polygon turning (G50.2) (T series)
- Balance cutting (G68) (T series (2-path))
- Cs contouring control
- Rigid tapping
- Spindle positioning (T series)

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 (T series (2-path)) and superimposed control (T series (2-path))
- 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 (T series)
- Rigid tapping

Signal

Program restart signal SRN<Gn006.0>

[Classification] Input signal

[Function] Selects program restart.

[Operation] When the program restart signal is set to "1" to search for the sequence number of the block to be restarted, the CRT screen changes to the program restart screen. When the program restart signal is set to "0", and automatic operation is activated, the tool is moved back to the machining restart point at dry run speed along the axes one by one in the sequence specified in parameter No. 7310. When the tool is set to the restart point, machining restarts.

Program restart under way signal SRNMV<Fn002.4>

[Classification] Output signal

[Function] Indicates the program is being restarted.

[Output cond.] The program restart under way signal becomes "1" when:

- The program restart signal is set to "0" after the CRT screen changes to the program restart screen.

The signal is set to "0" when:

- The program restart sequence ends (the tool has been moved to the restart point on all controlled axes).

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn006								SRN
Fn002				SRNMV				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7300	MOU	MOA						

[Input type] Parameter input

[Data type] Bit path

#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 MOU parameter 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								ROF

[Input type] Parameter input

[Data type] Bit path

#0 ROF When the coordinates for restarting are displayed on the program restart screen:

0: Tool length compensation (M series), tool position compensation (T series), cutter compensation (M series), and tool-nose radius compensation (T series) are considered.

1: Whether these compensation values are considered depends on the settings of bits 7 and 6 of parameter No. 3104 and bit 1 of parameter No. 3129 (parameters for specifying whether to consider each compensation value).

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
8135						NSQ		

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#2 NSQ Program restart is:

0: Used.

1: Not Used.

Alarm and message

Number	Message	Description
PS0060	SEQUENCE NUMBER NOT FOUND	A specified program restart sequence number cannot be found.
PS0071	DATA NOT FOUND	A specified program restart block number cannot be found.
PS0094	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.
PS0095	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.
PS0096	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.
PS0097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be directed when the program is restarted. (After power ON or alarms PS0094 to 0097 reset, no automatic operation is performed.) Perform automatic operation.
PS0098	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.

Number	Message	Description
PS0099	MDI EXEC NOT ALLOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI.
SR5020	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.

Warning

⚠ 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

⚠ CAUTION

1 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.

⚠ CAUTION
2 The M, S, T, and B (second auxiliary function) codes specified in the over store mode are not displayed on the program restart screen.
3 In the over store mode, the over store mode is not cancelled even though the operation mode is changed to a mode other than the MEM mode or RMT mode, but a value cannot be input in the over store item.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Program restart

5.8 RETRACE (M SERIES)

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.

NOTE
This function is optional.

Procedure

- Forward execution → reverse execution

To perform forward execution of a program, set the reverse execution signal RVS to 0, then perform a cycle start operation. If the reverse execution signal RVS is set to 1, reverse execution or the end of reverse execution results.
To perform reverse execution of a program, use one of the following methods:
1) Set the reverse execution signal RVS to 1 during forward execution of a block.
2) Perform a single block stop operation during forward execution, then set the reverse execution signal RVS to 1.
3) Perform a feed hold stop operation during forward execution, then set the reverse execution signal RVS to 1.

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 execution signal RVS is set to on.

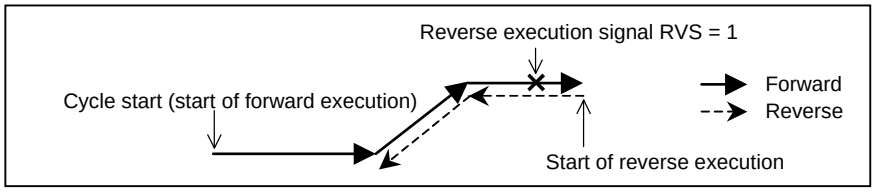


Fig. 5.8 (a)

When method 2) is used, performing a cycle start operation starts reverse execution from the position at which a single block stop takes place.

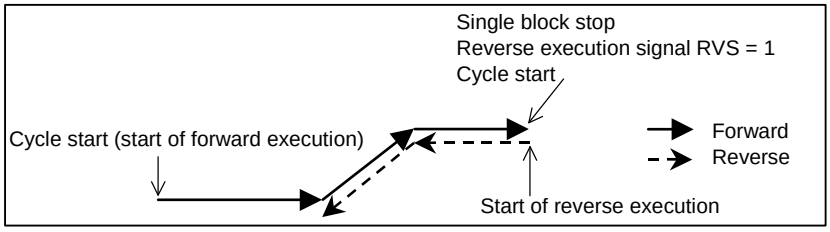


Fig. 5.8 (b)

When method 3) is used, performing a cycle start operation starts reverse execution from the position at which a feed hold stop takes place.

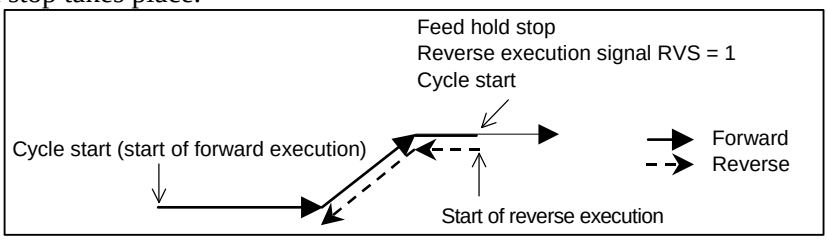


Fig. 5.8 (c)

- Reverse execution → forward reexecution

To perform forward reexecution of a program, use one of the following three methods:

1) Set the reverse execution signal RVS to 0 during reverse execution of a block.
2) Set the reverse execution signal RVS to 0 after a single block stop takes place during reverse execution.
3) Set the reverse execution signal RVS to 0 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 execution signal RVS is set to 0.

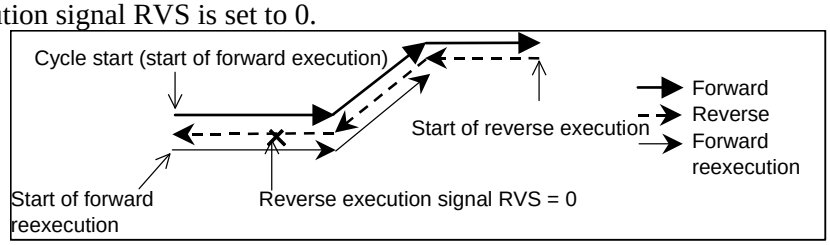


Fig. 5.8 (d)

When method 2) is used, performing a cycle start operation starts forward reexecution from the position at which a single block stop takes place.

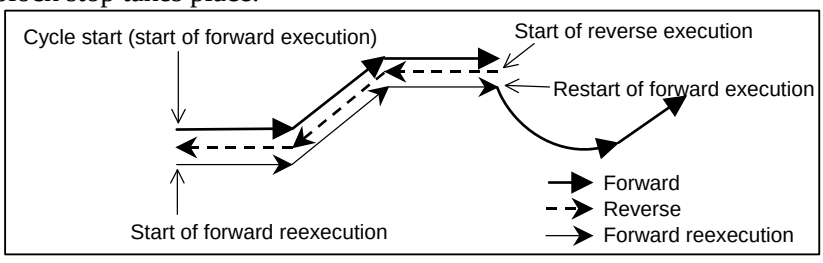


Fig. 5.8 (e)

When method 3) is used, performing a cycle start operation starts forward reexecution from the position at which a feed hold stop takes place.

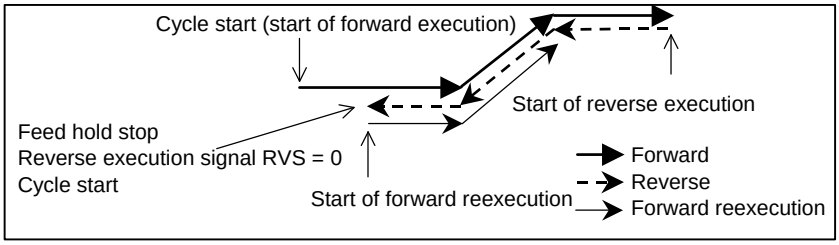


Fig. 5.8 (f)

- 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 execution signal RVS is held 1, operation is not performed, and the reverse execution end state is maintained. Forward reexecution (or forward execution) is started by setting the reverse execution signal RVS to 0 then performing a cycle start operation.

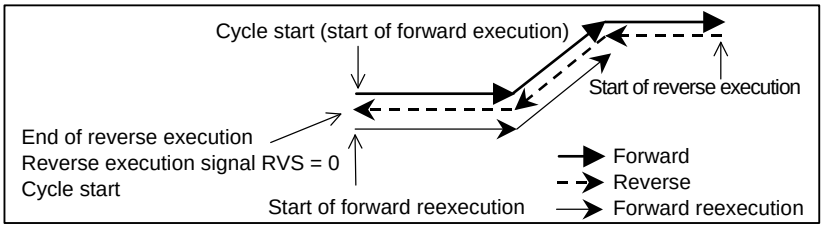


Fig. 5.8 (g)

- 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.

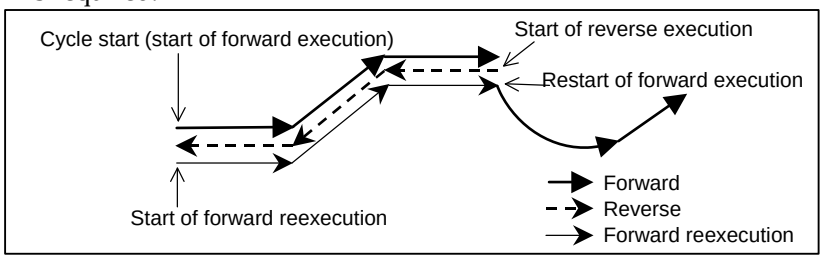


Fig. 5.8 (h)

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.

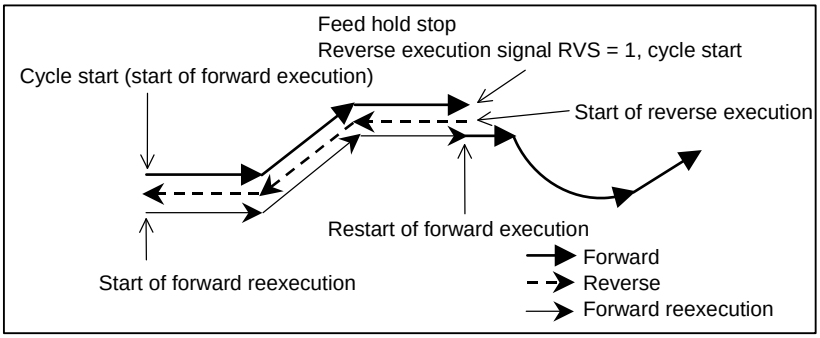


Fig. 5.8 (i)

Description

- 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 

key on the MDI panel, the external reset signal, or the reset & wind 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.

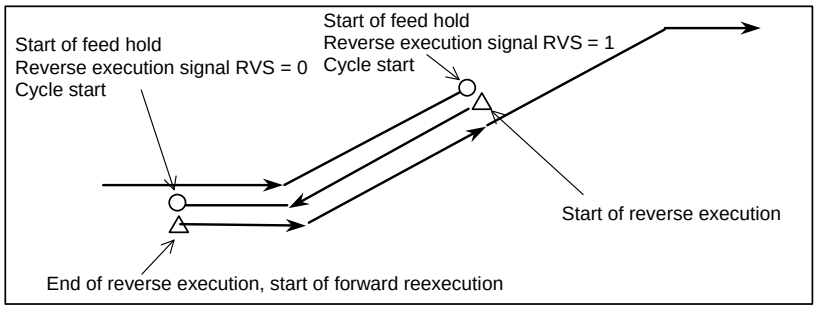


Fig. 5.8 (j)

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.

- Cylindrical interpolation (G07.1,G107)
- Polar coordinate command (G16)
- Thread cutting (G33)
- 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 (G05.1)
- HRV3 on/off (G05.4)
- Inch/metric conversion (G20, G21)
- Stored stroke check on/off (G22, G23)
- Functions related reference position return (G27, G28, G29, G30)
- Index table indexing
- Cs contouring control

- Switching of automatic operation mode

When a single block stop operation is performed during reverse execution or forward reexecution and the mode is changed from memory operation to MDI operation or vice versa, it becomes impossible to perform reverse execution, forward reexecution, and forward execution. To restart operation, restore the original mode, then perform a cycle start operation.

- Single block stop position

A block that is internally generated by the control unit is also treated as one block during reverse execution.

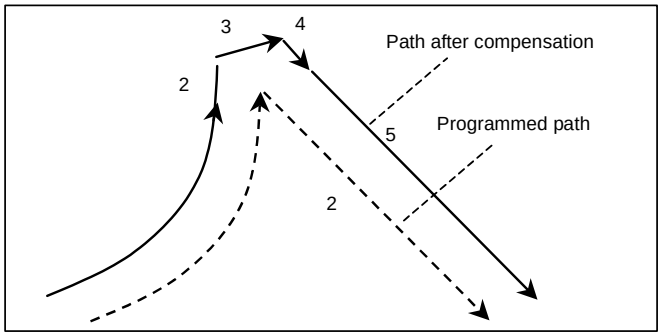


Fig. 5.8 (k) Path when cutter compensation is applied

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.

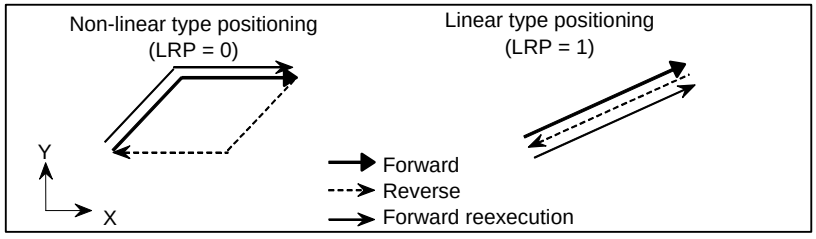


Fig. 5.8 (l)

- 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.

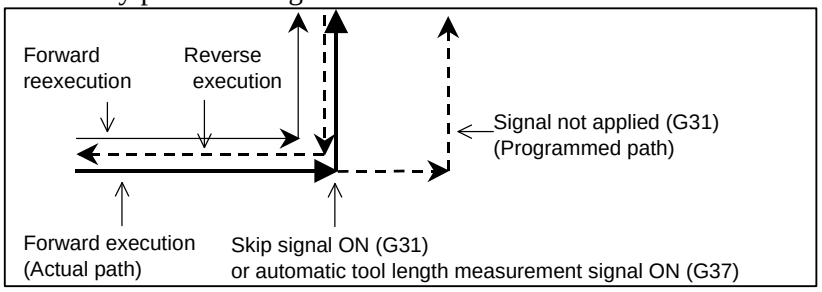


Fig. 5.8 (m)

- 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/AI contour control II

During reverse execution and forward reexecution, the feedrate clamp function by acceleration under AI contour control/AI contour control II 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.

Signal

Reverse execution signal RVS<Gn007.0>

[Classification] Input signal

[Function] Requests the control unit during automatic operation (DNC operation and MDI operation except memory operation and binary operation) to return the tool by retracing the path the tool has passed so far.

[Operation] When this signal is set to 1 during forward execution, the machining path along which the tool has moved so far is retraced. Note that, however, reverse execution does not start as soon as this signal is set to 1; reverse execution starts after the forward execution of the current block ends. When this signal is set to 0 during reverse execution, switching from reverse execution to forward reexecution takes place. Also in this case, forward reexecution does not start as soon as the signal is set to 0, but forward reexecution starts after the reverse execution of the current block is completed.
To make swift switching from forward execution to reverse execution or from reverse execution to forward reexecution, set the automatic operation stop signal *SP to 0 to stop automatic operation, and after the automatic operation in-progress signal STL becomes 0 and the automatic operation stop state is entered, change the RVS signal status. Subsequently, after the automatic operation stop signal *SP is set to 1 and the status of the automatic operation in-progress signal ST is changed from 1 to 0 to start automatic operation, it becomes possible to change the execution mode to reverse execution or forward reexecution in the middle of a block.

Reverse execution in-progress signal RVSL<Fn082.2>

[Classification] Output signal

[Function] Posts that reverse execution is being performed.

[Output cond.] This signal is set to 1 when:

- The reverse execution signal RVS is set to 1 and reverse execution is being performed.

This signal is set to 0 when:

- The reverse execution signal RVS is set to 0 and forward reexecution or forward execution is being performed.
- Blocks to be executed are no longer present during reverse execution, and so execution is stopped.

During reverse execution, the M, S, T, and second auxiliary functions are executed in the same way as during forward execution. When these functions are not to be performed in the same way as during forward execution, use this signal to allow the PMC to perform appropriate processing.
In particular, when the M, S, T, or second auxiliary function and a move command are specified in the same block, the positions at which the code signal and strobe signal are output differ between forward execution and reverse execution. Therefore, use this signal and distribution completion signal DEN to allow the PMC to perform appropriate processing as necessary.

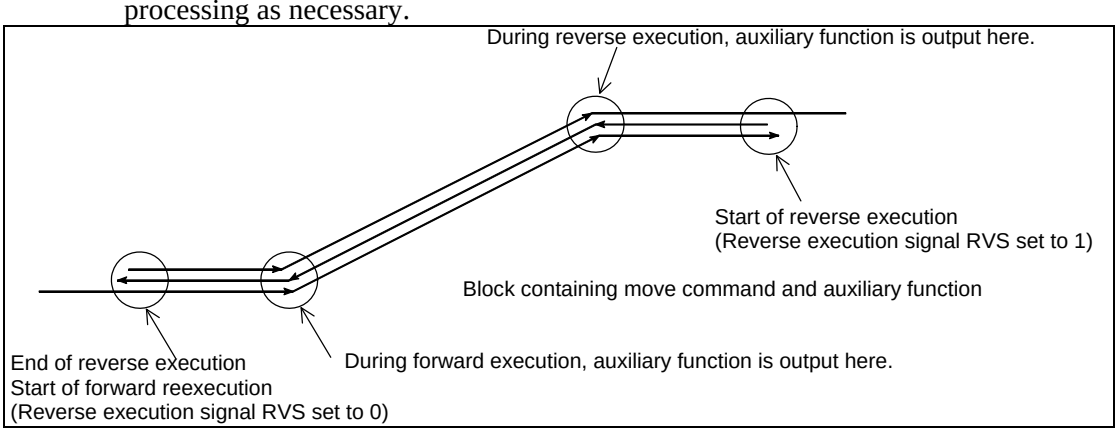


Fig. 5.8 (n)

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn007								
								RVS
	#7	#6	#5	#4	#3	#2	#1	#0
Fn082								
						RVSL		

Parameter

1414	
	Feedrate for retrace

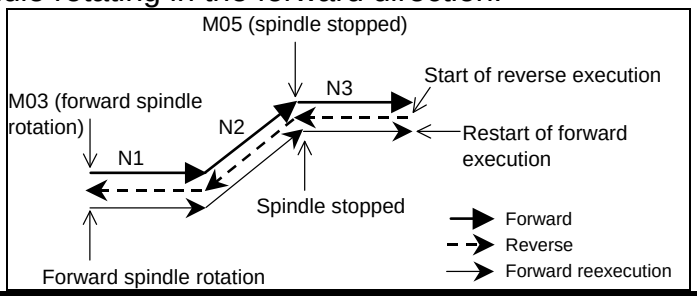
[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set a cutting feedrate for retrace operation of Retrace function. 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.

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.



⚠ WARNING

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 execution signal to 1. 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 signal *ABSM <Gn006.2>.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (M series) (B-64304EN-2)	Retrace

5.9 EXACT STOP / EXACT STOP MODE / TAPPING MODE / CUTTING MODE

Overview

NC commands can be used to control a feedrate in continuous cutting feed blocks as described below.

- Exact stop (G09)

The tool is decelerated in a block specifying G09, and an in-position check (*1) is performed. When the feed motor falls in-position, the tool is moved by the next block. This function may be used to produce a sharp edge at the corner of a workpiece.

- Exact Stop Mode (G61)

When G61 is commanded, deceleration of cutting feed command at the end point and in-position check is performed per block thereafter. This G61 is valid until G62 (automatic corner override), G63 (tapping mode), or G64 (cutting mode), is commanded.

- Tapping Mode (G63)

When G63 is commanded, feed rate override is ignored (always regarded as 100%), and feed hold also becomes invalid. Cutting feed does not decelerate at the end of block to transfer to the next block. This G63 is valid until G61 (exact stop mode), G62 (automatic corner override), or G64 (cutting mode) is commanded.

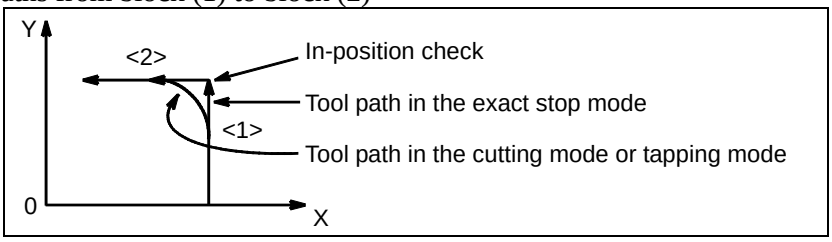
- Cutting Mode (G64)

When G64 is commanded, deceleration at the end point of each block thereafter is not performed and cutting goes on to the next block. This command is valid until G61 (exact stop mode), G62 (automatic corner override), or G63 (tapping mode) is commanded. However, in G64 mode, feed rate is decelerated to zero and in-position check is performed in the following case;

1) Positioning mode (G00, G60)
2) Block with exact stop check (G09)
3) Next block is a block without movement command

*1 The term in-position indicates that the servo motor reaches in a range of positions specified by a parameter. See Section “In-position check” and “In-position check independently of feed/rapid traverse” for details.

(Example) Tool paths from block (1) to block (2)



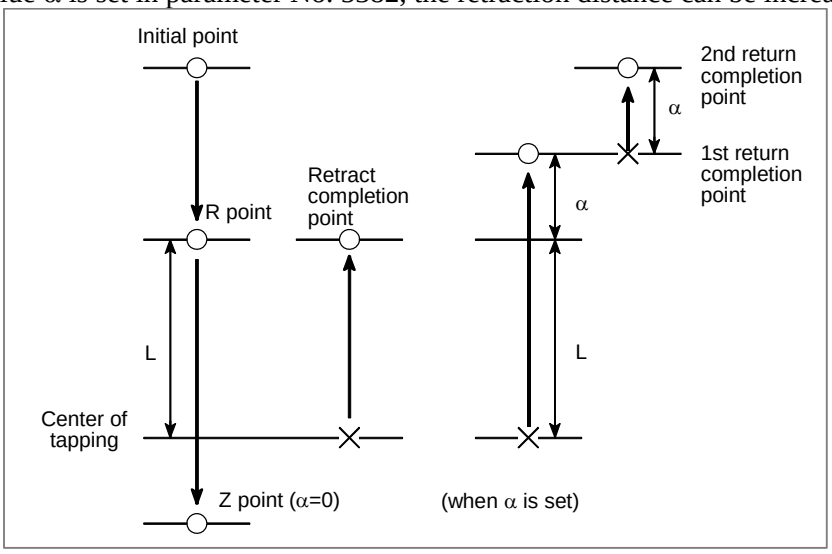
Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64304EN)	Exact stop (G09,G61) Cutting mode (G64) Tapping mode (G63)

5.10 RETRACTION FOR RIGID TAPPING

Overview

When rigid tapping is stopped, either as a result of an emergency stop or a reset, the tap may cut into the workpiece. The tap can subsequently be drawn out by using a PMC signal. This function automatically stores information relating to the tapping executed most recently. When a tap retraction signal is input, the tap is removed from the hole, based on the stored information. The tap is pulled toward the R point. When a retract value α is set in parameter No. 5382, the retraction distance can be increased by α.



Basic procedure

(1) Start

Reset the CNC, then select MDI mode. Setting rigid tapping retraction start signal RTNT to "1" starts rigid tapping retraction.

(2) Completion

Upon the completion of rigid tapping retraction, rigid tapping retraction completion signal RTPT is set to "1", with which the CNC automatically enters the reset state. Setting rigid tapping retract start signal RTNT to "0" sets rigid tapping retraction completion signal RTPT to "0".

(3) Stop
During rigid tapping retraction, setting rigid tapping retraction start signal RTNT to "0" stops rigid tapping retraction, placing the CNC in the reset state. To resume rigid tapping retraction, set rigid tapping retraction start signal RTNT to "1". Rigid tapping retraction can also be stopped by means of a reset or feed hold.

(4) Resume
Once rigid tapping retraction has been stopped, it can be resumed by performing the same operation as that used for starting rigid tapping retraction. If rigid tapping retraction has been completed, however, the start operation does not restart rigid tapping retraction. If retract value α is set in parameter No. 5382, however, the start operation performs rigid tapping retraction using α only.

- Start and completion time chart

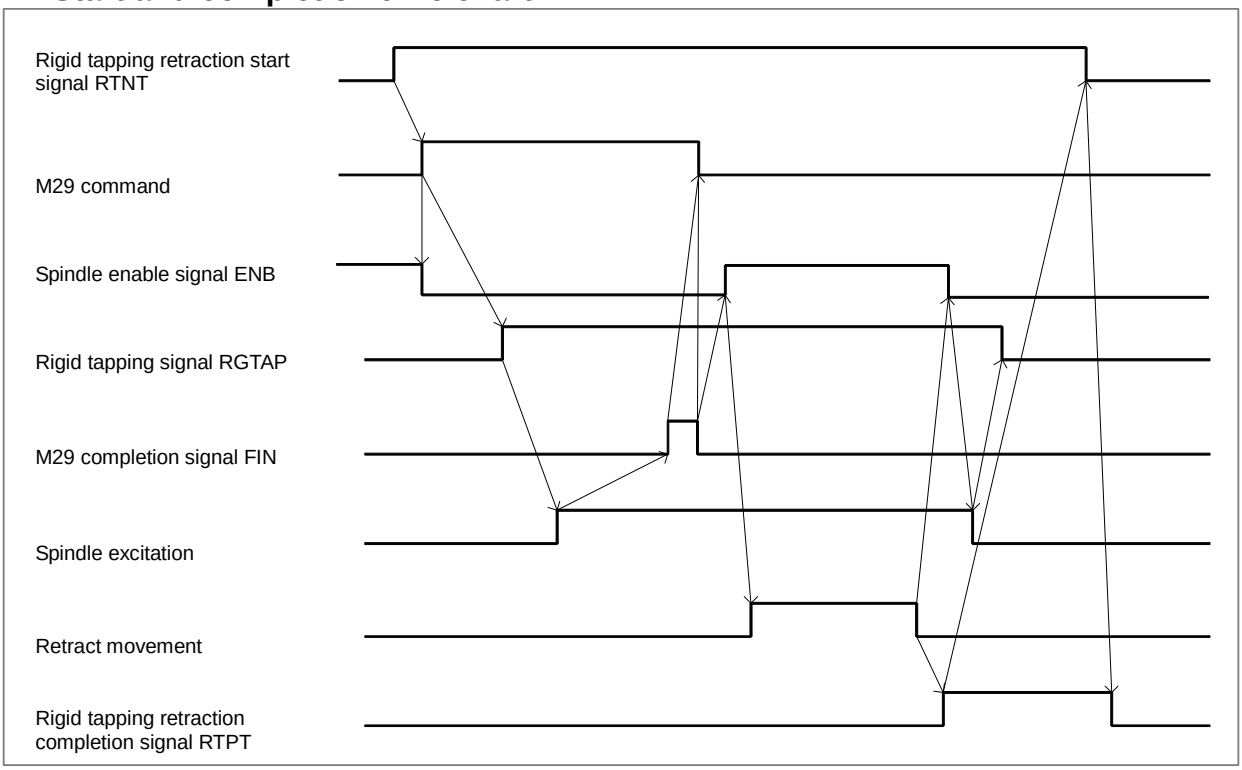
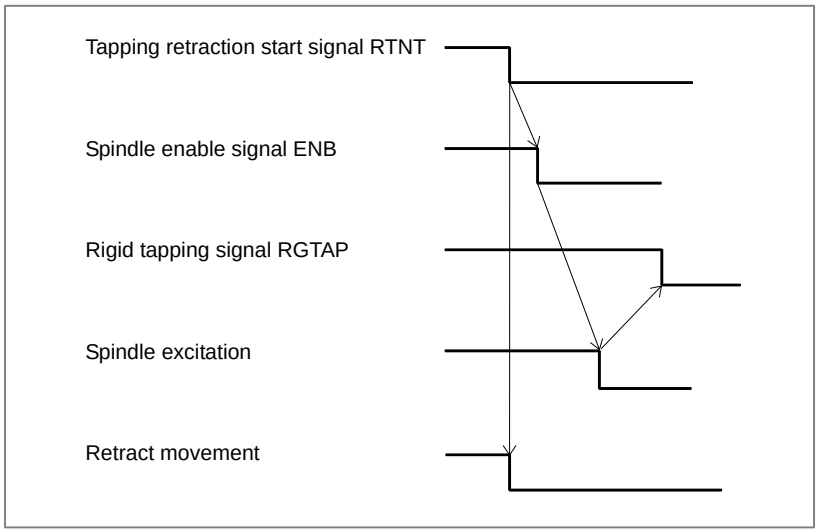


Fig. 5.10 (a) Start and completion time chart

In the reset state, setting rigid tapping retraction start signal RTNT to "1" in MDI mode causes the rigid tapping M command to be output. For rigid tapping retraction, specify neither gear switching nor orientation. Spindle function strobe signal SF is also output if no S command has been specified after power-on.
Upon the completion of rigid tapping retraction, spindle enable signal ENB is set to "0", in the same way as at the end of ordinary rigid tapping. Therefore, perform the sequence for canceling rigid tapping. Once rigid tapping retraction has been completed, rigid tapping retraction completion signal RTPT is set to "1" and the CNC enters the reset state.

- **Time chart for stopping tapping retraction**



When tapping retraction is stopped, spindle enable signal is set to 0, in the same way as for ordinary rigid tapping. Therefore, perform the sequence for canceling rigid tapping. The CNC also automatically enters the reset state when tapping retraction is stopped.

Rigid tapping retraction using the G30 command

The tap can also be drawn out by setting the relevant G30 command parameter in the program. Even if the power supply is interrupted during rigid tapping due to a power failure or for some other reason, this function allows the tap cut into the workpiece to be drawn out, when the servo axis is the absolute position detector.

・Specification method

Rigid tapping retraction using the G30 command is possible when 1 is set in bit 1 (RG3) of parameter No. 5202.

If rigid tapping is stopped either as a result of a reset or emergency stop, or if the power supply is interrupted during rigid tapping, the tapping axis returns to the initial point or R point in synchronization with the spindle, based on the rigid tapping command information in the machining program, when the following command is executed in the MEM or MDI operation.

The rigid tapping retraction command is a one-shot code.

G30 P99 M29 S $\underline{\text{min}^{-1}}$;

・Time chart

The starting time chart is the same as that for regular rigid tapping.

Note that, since the G30 rigid tapping retraction command is a one-shot code, the G80 command is not available for canceling rigid tapping.

Therefore, in addition to the steps of the regular time chart for stopping rigid tapping, it is necessary to add a step for canceling the rigid tapping signal RGTAP<Gn061.0> using the rigid tapping retraction completion signal RTPT<Fn066.1>.

・Limitations

Rigid tapping retraction using the G30 command (1 is set in bit 1 (RG3) of parameter No. 5202) has the following limitations.

1 Rigid tapping retraction cannot be performed using the input signal RTNT<Gn062.6>.

2 Bit 0 (G84) of parameter No. 5200 must be set to 0.

(Example)

Machining program

M29 S1000 ;

```
G84 X20.0 Y20.0 R-10.0 Z-30. F500 ;
X50.0 Y50.0 ;
X100.0 Y100.0 ;
G80
----------

Retraction program
----------
G30 P99 M29 S1000 ;
G00 Z-10.0 ;
----------
```

NOTE
If a value other than 0 is set in parameter No. 5210, specify that value instead of M29 in the above program.

Limitation

T

In case of G code system A for a lathe system, rigid tapping retraction cannot be used when the drilling axis of the tapping is an axis parallel to one of the basic three axes.

Signal

Rigid tapping retraction start signal RTNT<Gn062.6>

[Classification] Input signal
[Function] Starts rigid tapping retraction.
[Operation] When this signal is set to "1", the control unit operates as follows:
- Starts rigid tapping retraction.

Rigid tapping retraction completion signal RTPT<Fn066.1>

[Classification] Output signal
[Function] Notifies the completion of rigid tapping retraction.
[Output cond.] This signal is set to "1" in the following case:
- Rigid tapping retraction has been completed.

This signal is set to "0" in the following case:
- Rigid tapping retraction start signal has been set to "0".

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn062		RTNT						

	#7	#6	#5	#4	#3	#2	#1	#0
Fn066							RTPT	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5200				DOV				

[Input type] Parameter input
[Data type] Bit path

#4 DOV Override during extraction in rigid tapping
0: Invalidated
1: Validated (The override value is set in parameter No.5211. However, set an override value for rigid tapping return in parameter No. 5381.)

	#7	#6	#5	#4	#3	#2	#1	#0
5202							RG3	

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#1 RG3 A rigid tapping return operation is specified:
0: With input signal RTNT <G062.6>.
1: With one-shot G code G30.

NOTE
To use this function, set bit 0 (G84) of parameter No. 5200 to 0.

5381	Override value during rigid tapping return

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 0 to 200
This parameter is used to set the override value during rigid tapping return.
If the setting is 0, no override is applied.

NOTE
This parameter is valid when bit 4 (DOV) of parameter No. 5200 for enabling override at normal extraction time is set to 1.

5382	Amount of return for rigid tapping return

[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 drilling 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 is used to set an extra amount of rigid tapping return. The tool is retracted additionally near point R by the distance set in this parameter. If the tool has already been retracted from rigid tapping, it will be retracted further only by the distance specified in this parameter.

Caution

⚠ CAUTION

1 If rigid tapping is stopped as a result of an emergency stop, the position on the tapping axis (Z-axis) is maintained but the spindle position is lost. In such a case, therefore, the positional relationship between the spindle and tapping axis is not guaranteed when operation is resumed.
2 Rigid tapping retraction is performed based on the tapping axis (Z-axis) commands accumulated for tapping. If rigid tapping is stopped as a result of an emergency stop, therefore, rigid tapping retraction may fail to draw the tapping tool completely out of the workpiece. In such a case, set retract value α (parameter No. 5382).
3 During rigid tapping retraction, switching the mode to the manual operation mode stops rigid tapping retraction, and decelerates and stops the machine.
4 For rigid tapping retraction, the CNC internally activates a return program. Rigid tapping retraction may, therefore, cause some G codes or M/F/S codes to be overwritten (G80/G84/G74, G94/G95).
5 During rigid tapping retraction, do not change any setting affecting the travel distance (such as the increment system, input unit 10 time multiply, or calculator-type decimal notation) that was made when machining data was stored for rigid tapping retraction.

Note

NOTE

1 Setting rigid tapping retraction start signal RTNT to 1 starts rigid tapping retraction only when the CNC is placed in both the reset state and MDI mode.
2 The machining data for rigid tapping retraction is retained until the next rigid tapping command is specified, even after the power is turned off. Rigid tapping retraction can, therefore, be specified even after the power is turned off.
3 Rigid tapping retraction is not performed if the input increment (inches or millimeters) selected when machining data is stored for rigid tapping retraction differs from that selected when rigid tapping retraction is executed.
4 Override can be applied to rigid tapping retraction when it is enabled.

5.11 DNC OPERATION

Overview

When automatic operation is performed in the DNC operation mode (RMT), machining (DNC operation) can be performed while a program is being read via the reader/puncher interface or from a memory card. Before performing DNC operation, set parameters relating to the reader/puncher interface.

Signal

DNC operation select signal DNCI<Gn043.5>

[Classification] Input signal

[Function] Selects the DNC operation mode (RMT).
To select the DNC operation mode (RMT), the memory operation mode (MEM) must be selected, and the DNC operation selection signal must be set to 1 at the same time.

[Operation] When this signal is set to 1, the control unit operates as follows:
- If the memory operation mode (MEM) is not selected, the control unit ignores this signal and does nothing.

- If the memory operation mode (MEM) is selected, the DNC operation mode (RMT) is selected, enabling DNC operation. At this time, DNC operation selection confirm signal MRMT is set to 1.

DNC operation selection confirm signal MRMT<Fn003.4>

[Classification] Output signal
[Function] Indicates that the DNC operation mode (RMT) is selected.
[Output cond.] This signal is set to 1 when:
- The DNC operation mode (RMT) is selected.

This signal is set to 0 when:
- Selection of the DNC operation mode (RMT) ends.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn043			DNCI					
Fn003				MRMT				

Parameter

	#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. (A compact flash card adapter is required separately.)

NOTE
1 The compact flash card must be inserted in the front PCMCIA slot by using a compact flash card adapter.
2 When a DNC operation using a memory card is being performed, the memory card cannot be accessed to, for example, display a list of data stored on the memory card.
3 DNC operation using a memory card cannot be performed when the system has two paths.
4 During DNC operation using a memory card, never remove the memory card.
5 A program being executed by DNC operation cannot call another program on the memory card.
6 The memory card utilities of the CNC screen display function cannot be used to perform DNC operation.

	#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

> ⚠ **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.

Alarm and message

Number	Message	Description
SR0086	DR OFF	During I/O process by reader/puncher interface 1, the data set ready input signal of the I/O device (DR) was 0. Possible causes are an I/O device not turn on, a broken cable, and a defective printed circuit board.
SR1830	DR OFF(2)	During I/O process by reader/puncher interface 2, the data set ready input signal of the I/O device (DR) was 0. Possible causes are an I/O device not turn on, a broken cable, and a defective printed circuit board.
PS0123	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.
PS1081	EXT DEVICE SUB PROGRAM CALL MODE ERROR	The external device subprogram call is not possible in this mode.

5.12 MANUAL INTERVENTION AND RETURN

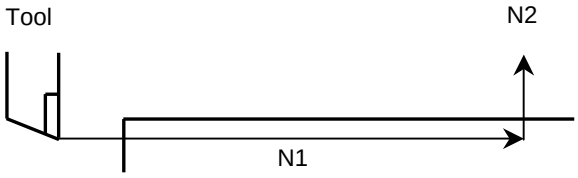
Overview

When you perform a restart after stopping the axis movement by feed hold during automatic operation and intervening in the tool position manually for such purposes as to check the cutting surface, this function automatically returns the tool to the position where it was prior to manual intervention, before resuming automatic operation.

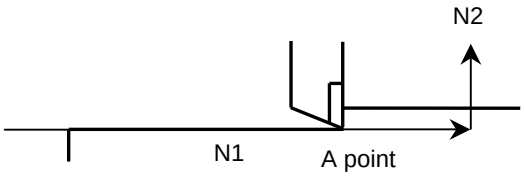
Explanation

The manual intervention and return function is enabled by setting 1 in bit 0 (MIT) of parameter No. 7001. The manual intervention and return sequence is as follows.

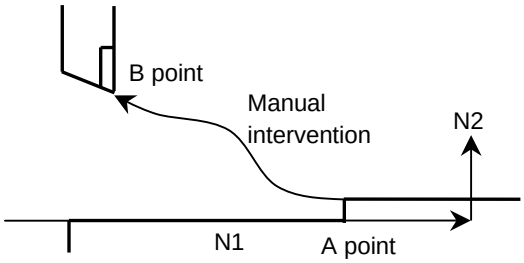
1. Cut the N1 block.



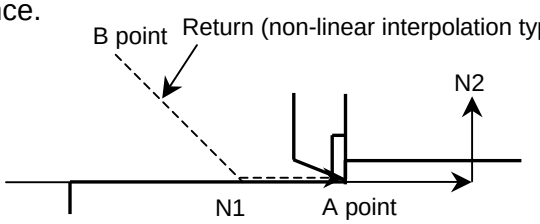
2. Stop movement by feed hold while cutting the N1 block (A point).



3. Move the tool to the B point manually, and restart the automatic operation.

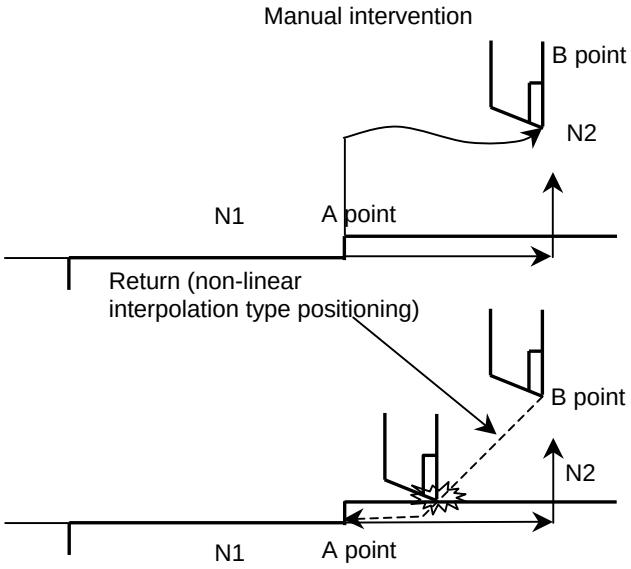


4. After the tool returns to the A point by non-linear interpolation type positioning at dry run federate, the command is executed to cause the N1 block to travel the remaining distance.



⚠ WARNING

When intervening manually, exercise due care to handle the tool in a manner appropriate for the machining direction and the object being machine; otherwise, the object, machine, and/or tool may be damaged.



・Manual absolute on/off

In the manual absolute on state, the tool behaves as specified by the manual absolute on/off function, instead of returning to the position where it stopped. For details, see "MANUAL ABSOLUTE ON/OFF" earlier in this manual.

・Return feedrate

The return feedrate is the dry run feedrate, and the jog feedrate override function is valid. When the manual rapid traverse selection signal (RT<Gn019.7>) is set to "1", the return feedrate is the rapid traverse feedrate, instead of the dry run feedrate.

・Return operation

The return operation uses non-linear type positioning.

・Single block

When the single block stop switch is on at the time of return, the tool stops temporarily at the stop position and resumes operation at the next cycle start.

・Cancellation

If a reset, alarm, or emergency stop occurs during manual intervention or return, the manual intervention and return function is canceled.

・MDI mode

The manual intervention and return function is also valid in MDI mode.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7001								MIT

[Input type] Parameter input
[Data type] Bit path

#0 MIT The manual intervention and return function is:
0: Disabled.
1: Enabled.

Limitations

- When the manual intervention and return function becomes valid or invalid

The function is valid only if manual intervention occurs when the automatic operation stop lamp is lit. If you stop the tool by feed hold when the remaining distance is 0 and intervene manually, the manual intervention and return function becomes invalid and the tool behaves as specified by the manual absolute on/off function. For details, see "MANUAL ABSOLUTE ON/OFF" earlier in this manual.

- Offset

If you change the offset value when you resume the operation in the middle of a block where the operation was stopped after replacing the tool manually due to tool breakage or for some other reason, the changed offset is not reflected on machining.

- Machine lock, mirror image, and scaling

Do not use machine lock, mirror image, or scaling when performing the manual intervention and return function.

5.13 DIRECT OPERATION BY C LANGUAGE EXECUTOR

Overview

When the DNC operation mode (RMT) is set, and automatic operation is started, it becomes possible to perform machining (direct operation = DNC operation) while reading a program from the C language executor.

For details, refer to the "C Language Executor Programming Manual (B-64303EN-3)".

Signal

Direct operation select signal DMMC<Gn042.7>

[Classification] Input signal

[Function] Selects the mode (direct operation mode) for performing machining while reading a program from the C language executor.

[Operation] When this signal is set to 1, the control unit operates as follows:

- If the memory operation mode (MEM) is not selected, the control unit ignores this signal and does nothing.
- If the memory operation mode (MEM) is selected, the direct operation mode is selected, enabling direct operation.

DNC operation selection confirm signal MRMT<Fn003.4>

[Classification] Output signal

[Function] Posts that the DNC operation mode (RMT) is selected.

[Output cond.] This signal is set to 1 when:

- The DNC operation mode (RMT) is selected.

This signal is set to 0 when:

- Selection of the DNC operation mode (RMT) ends.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn042	DMMC							
Fn003				MRMT				

6 INTERPOLATION FUNCTION

Chapter 6, "INTERPOLATION FUNCTION", consists of the following sections:

6.1 POSITIONING 426
6.2 SINGLE DIRECTION POSITIONING (M SERIES) 427
6.3 LINEAR INTERPOLATION 430
6.4 CIRCULAR INTERPOLATION 432
6.5 THREADING 435
6.6 HELICAL INTERPOLATION 448
6.7 POLAR COORDINATE INTERPOLATION (T SERIES) 449
6.8 CYLINDRICAL INTERPOLATION 452
6.9 POLYGON TURNING (T SERIES) 454
6.10 NORMAL DIRECTION CONTROL (M SERIES) 480
6.11 LINEAR INTERPOLATION (G28, G30, G53) 483
6.12 NANO SMOOTHING (M SERIES) 484
6.13 GENERAL PURPOSE RETRACT 491

6.1 POSITIONING

Overview

The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental command at a rapid traverse rate. In the absolute command, coordinate value of the end point is programmed. In the incremental command the distance the tool moves is programmed.
The tool path is determined by selecting one of the following with parameter LRP (No. 1401#1):

- Linear interpolation type positioning
  The tool is positioned using a straight path and a speed that is not higher than the rapid traverse of each axis but that assures the shortest positioning time. By changing the acceleration/ deceleration type from the constant acceleration/deceleration (inclination) type to the constant time (time constant) type with parameter PRT (No. 1603#4), the tool can be moved along a specified path.
- Non-linear interpolation type positioning
  Positioning is performed with each axis independently at the rapid traverse rate. Generally, the tool path is not a straight line.

The rapid traverse rate in the 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.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401							LRP	

[Input type] Parameter input
[Data type] Bit path

#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.

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.

	#7	#6	#5	#4	#3	#2	#1	#0
1603				PRT				

[Input type] Parameter input
[Data type] Bit path

#4 PRT For positioning of linear interpolation type:
0: Acceleration/deceleration of acceleration fixed type is used.
1: Acceleration/deceleration of time fixed type is used.

Note

NOTE
The rapid traverse rate cannot be specified in the address F.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Positioning (G00)

6.2 SINGLE DIRECTION POSITIONING (M SERIES)

M

Overview

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

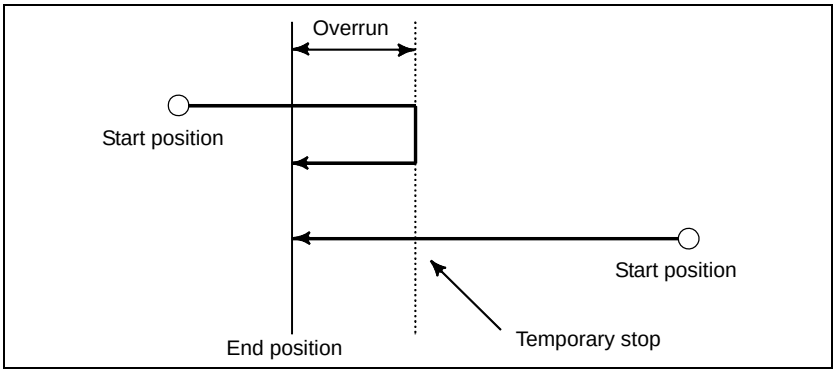


Fig.6.2 (a) Direction positioning process

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.

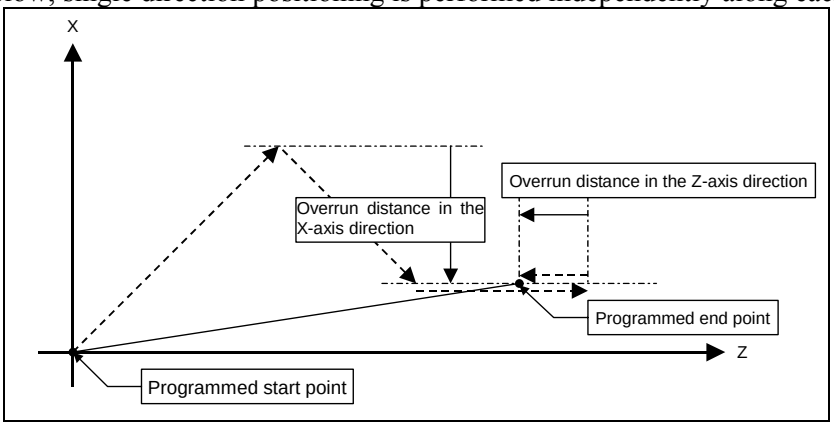
G60, which is an one-shot G-code, can be used as a modal G-code in group 01 by setting 1 to the parameter (No. 5431 bit 0 G60).
This setting can eliminate specifying a G60 command for every block. Other specifications are the same as those for an one-shot G60 command.
When an one-shot G code is specified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

(Example)

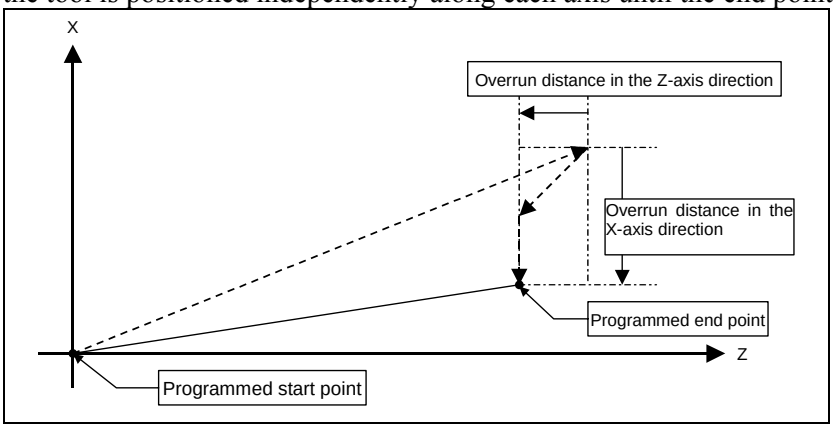
When one-shot G60 commands are used.		When modal G60 command is used.	
:			:
:			
:			
G90;			G90 G60; Single direction positioning mode start
direction			
G60 X0 Y0;		X0 Y0;	
G60 X100;	Single direction	X100;	Single direction
G60 Y100;	positioning	Y100;	positioning
G04 X10;		G04 X10;	
G00 X0 Y0;		G00 X0 Y0;	Single direction

- Overview of operation

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



- **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.



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5431								
							PDI	MDL

[Input type] Parameter input
[Data type] Bit

#0 MDL Specifies whether the G60 code (single direction positioning) is a one-shot G code (00 group) or modal G code (01 group).
0: One-shot G code (00 group)
1: Modal G code (01 group)

#1 PDI In the G60 mode, at a temporary stop position, an in-position check is:
0: Not made. (Only the end of acceleration/deceleration is awaited.)
1: Made.

5440	
	Overrun distance and direction of unidirectional positioning (G60)

[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)
For each axis, set the overrun distance and direction of unidirectional positioning (G60). The positioning direction is determined by the sign of the data set in this parameter, and the overrun distance is determined by the value of the data set in this parameter.

Overrun distance > 0: Positioning direction is positive (+).
Overrun distance < 0: Positioning direction is negative (-).
Overrun distance = 0: Unidirectional positioning is not performed.

Alarm And message

Number	Message	Contents
DS0025	G60 cannot be executed.	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, so that single direction positioning cannot be executed. Correct the program.

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.

- When specifying single direction positioning on a machine that uses 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.
- Single direction positioning cannot be performed for a hole machining axis in a canned cycle for hole machining.
- Single direction positioning is not performed for an axis subject to shift value movement in a G76 or G87 canned cycle.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (M series) (B-64304EN-2)	Single direction positioning

6.3 LINEAR INTERPOLATION

Overview

Tools can move along a line. 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 $\alpha\alpha$ $\beta\beta$ $\gamma\gamma$ $\zeta\zeta$;

Feed rate of α axis direction : $F\alpha=\alpha/L\times f$
Feed rate of β axis direction : $F\beta=\beta/L\times f$
Feed rate of γ axis direction : $F\gamma=\gamma/L\times f$
Feed rate of ζ axis direction : $F\zeta=\zeta/L\times f$

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

The feedrate of the rotary axis is commanded in the unit of deg/min (if the feedrate is 12 deg/min, F12.0 is commanded).

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 feed rate is that in which the tangential feed rate 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.

(Example)

G91 G01 X20.0 C40.0 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:

$$\frac{\sqrt{20^2+40^2}}{300}\fallingdotseq 0.14907\ \text{min}$$

The feed rate for the C axis is

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

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

Parameter

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)
When the machine requires little change in cutting feedrate during cutting, a cutting feedrate can be specified in the parameter. This eliminates the need to specify a cutting feedrate (F command) in the NC program.

The feedrate set in this parameter is valid from the time when the CNC enters the clear state (1 is set in bit 6 (CLR) of parameter No. 3402) due to powering on, reset, or other cause until a feedrate is specified by a program command (F command). When a feedrate is specified by the F command, that specified feedrate takes effect. For information about the clear state, refer to the appendix of the Operator's Manual (B-64304EN).

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.
In cutting feed, the maximum feedrate is clamped so that the feedrate of each axis does not exceed its maximum cutting feedrate.

	#7	#6	#5	#4	#3	#2	#1	#0
3402								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)

Alarm and message

Number	Message	Description
PS0011	FEED ZERO (COMMAND)	The cutting feedrate instructed by an F code has been set to 0. 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.

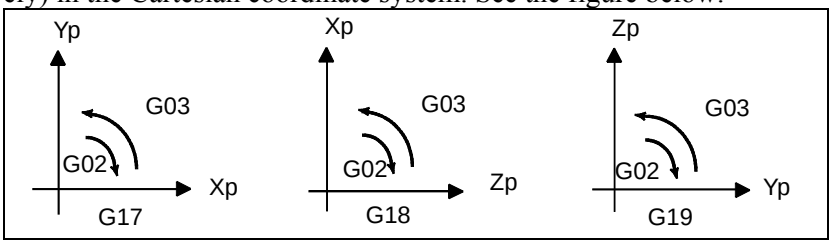
Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Linear interpolation (G01)

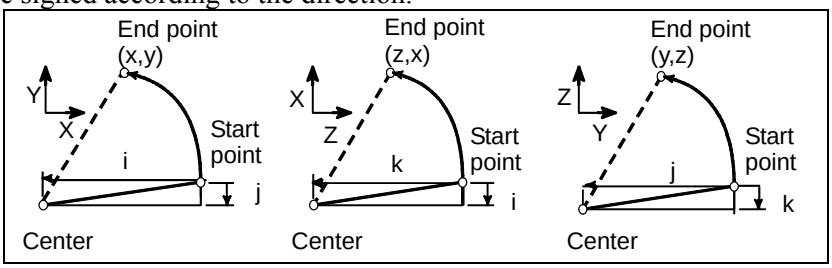
6.4 CIRCULAR INTERPOLATION

Overview

The command below can move a tool along a circular arc in the defined plane.
"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.



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 a sign.
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, as shown below.
I, J, and K must be signed according to the direction.



I0,J0, and K0 can be omitted. 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
If the difference between the radius at the start point and that at the end point exceeds the value in a parameter (No.3410), an alarm PS0020 occurs.
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.

M

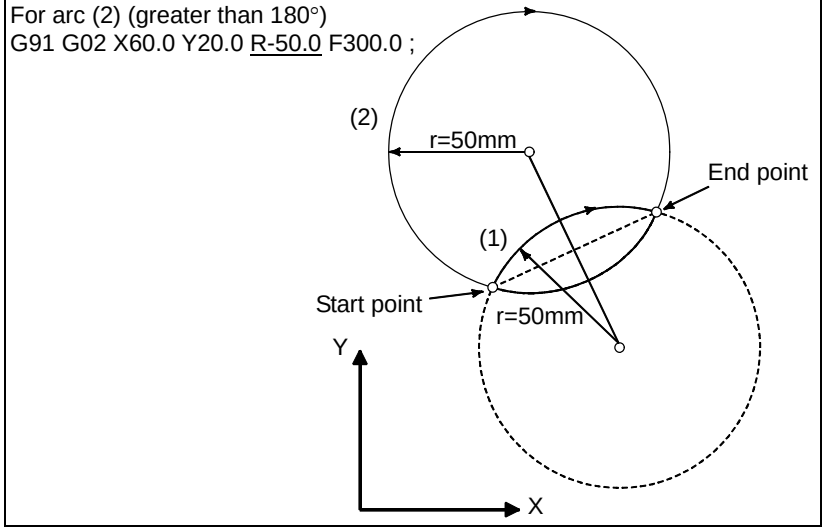
To specify an arc of greater than 180°, set the radius using a negative value.

T

It is not allowed to specify an arc of greater than 180° (the radius cannot be set using a negative value). Specifying such an arc causes an alarm PS0023.

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 05is programmed.
G02R_ ; (The tool does not move.)

(Example)
For arc (1) (less than 180°)
G91 G02 X60.0 Y20.0 R50.0 F300.0 ;
For arc (2) (greater than 180°)
G91 G02 X60.0 Y20.0 R-50.0 F300.0 ;



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.

Parameter

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 (M series), and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane, G19: Yp-Zp plane), 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	Rotation 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.

	#7	#6	#5	#4	#3	#2	#1	#0
3402								
						G19	G18	

[Input type] Parameter input
[Data type] Bit path

#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.

G19	G18	G17, G18, or G19 mode
0	0	G17 mode (X-Y plane)
0	1	G18 mode (Z-X plane)
1	0	G19 mode (Y-Z plane)

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.

NOTE
When the setting is 0, the difference between the arc radius values is not checked.

Alarm and message

Number	Message	Description
PS0011	FEED ZERO (COMMAND)	The cutting feedrate instructed by an F code has been set to 0. 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.
PS0020	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. 2410. Check arc center codes I, J and K in the program. The tool path when parameter No. 2410 is set to a large value is spiral.
PS0021	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. To specify three or more axes requires the helical interpolation option (T series).

Number	Message	Description
PS0023	ILLEGAL RADIUS COMMAND	A negative value is specified as the radius of the arc. In the T series, an arc of greater than 180° cannot be specified in the R command. Modify the program.
PS0025	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).
PS0028	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. To specify three or more axes requires the helical interpolation option (T series).

Note

NOTE

1 For T series, the U, V and W axes (parallel with the basic axis) can be used with G-code system B and C.
2 If I, J, K, and R addresses are specified simultaneously, the arc specified by address R takes precedence and the other are ignored.
3 If an axis not comprising the specified plane is commanded, an alarm is displayed.
For example, when G code system B or C is used, if U axis with X axis is specified as a parallel axis to X axis when plane XY is specified, an alarm PS0028 is displayed.

Reference item

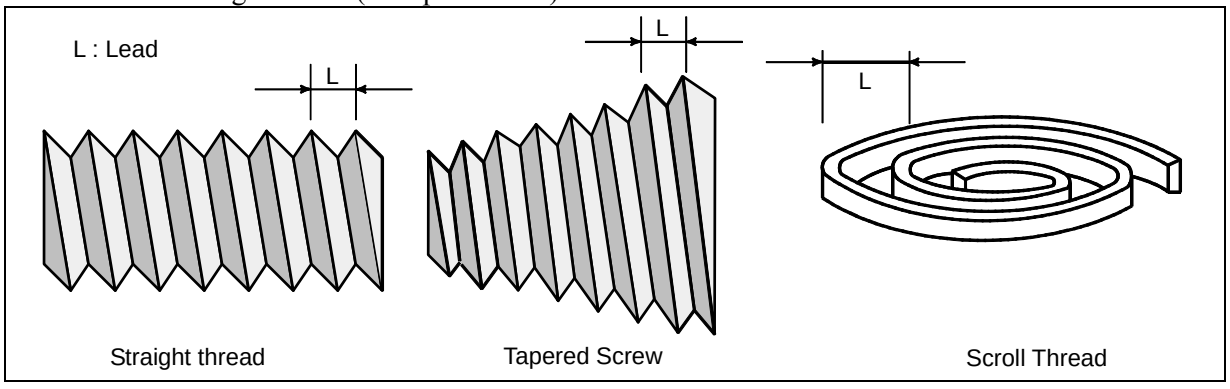
Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Circular interpolation (G02, G03)

6.5 THREADING

6.5.1 Threading

Overview

Tool movement can be synchronized with spindle rotation when cutting threads.
The spindle speed is continuously read through the position coder attached to the spindle. Then, it is converted to a cutting feedrate (feed per minute) to feed the tool.



In general, thread cutting is repeated along the same tool path in rough cutting through finish cutting for a screw. Since thread cutting starts when the position coder mounted on the spindle outputs a 1-turn signal, threading is started at a fixed point and the tool path on the workpiece is unchanged for repeated thread cutting. Note that the spindle speed must remain constant from rough cutting through finish cutting. If not, incorrect thread lead will occur.

Signal

Threading signal THRD<Fn002.3>

[Function] This signal indicates that threading is in progress.

[Output cond.] This signal turns to "1" in the following cases:

- Threading mode in progress
- Threading cycle for turning

This signal turns to "0" in the following case.

- Neither threading mode nor thread cutting cycle are in progress.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn002					THRD			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401			TDR					

[Input type] Parameter input

[Data type] Bit path

#5 TDR Dry run during threading or tapping (tapping cycle G74 or G84, rigid tapping)

0: Enabled

1: Disabled

	#7	#6	#5	#4	#3	#2	#1	#0
1402				JRV				

[Input type] Parameter input

[Data type] Bit path

#4 JRV Jog feed or incremental feed is

0: Performed at feed per minute.

1: Performed at feed per rotation.

NOTE
Specify a feedrate in parameter No.1423.

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 JRV, bit 4 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 JRV, bit 4 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).

	#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.

	Acceleration/deceleration time constant in threading cycles for each axis
1626	

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
Set a time constant for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis.

	FL rate for acceleration/deceleration in threading cycles for each axis
1627	

[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 an FL feedrate for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis. Set 0 at all times except in a special case.

	Cutting value (chamfering value) in thread cutting cycles G92 and G76
5130	

[Input type] Parameter input
[Data type] Byte path
[Unit of data] 0.1
[Valid data range] 0 to 127
This parameter sets a cutting value (chamfering value) in the thread cutting cycle (G76) of a multiple repetitive canned cycle and in the thread cutting cycle (G92) of a canned cycle.
Let L b a lead. Then, a cutting value range from 0.1L to 12.7L is allowed.
To specify a cutting value of 10.0L, for example, specify 100 in this parameter.

5140	Minimum depth of cut in the multiple repetitive canned cycle G76

[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 the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets a minimum depth of cut in the multiple repetitive canned cycle G76 so that the depth of cut does not become too small when the depth of cut is constant.

NOTE
Specify a radius value at all times.

5141	Finishing allowance in the multiple repetitive canned cycle G76

[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 the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the finishing allowance in multiple repetitive canned cycle G76.

NOTE
Specify a radius value at all times.

5142	Repetition count of final finishing in multiple repetitive canned cycle G76

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Cycle
[Valid data range] 1 to 99999999
This parameter sets the number of final finishing cycle repeats in the multiple repetitive canned cycle G76 or G76.7.
When 0 is set, only one final finishing cycle is executed.

5143	Tool nose angle in multiple repetitive canned cycle G76

[Input type] Parameter input
[Data type] Byte path
[Unit of data] Degree
[Valid data range] 0, 29, 30, 55, 60, 80
This parameter sets the tool nose angle in multiple repetitive canned cycle G76.
This parameter is not used with the Series 10/11 program format.

	#7	#6	#5	#4	#3	#2	#1	#0
11602			NCP					

[Data type] Bit path

#5 NCP If there is a non-threading block between two threading blocks, the second threading block:

0: Waits until the spindle one-rotation signal and the spindle speed arrival signal (SAR) are detected.

1: Does not wait until the spindle one-rotation signal and the spindle speed arrival signal (SAR) are detected unless a G code in non-threading group 01 is issued. (FS0i-C-compatible specification)

Warning

⚠ WARNING

During threading, stopping feed without stopping the spindle is dangerous because the cutting depth will abruptly increase. Feed hold is, therefore, disabled during threading. If attempted during threading, feed stops in the same way as single block stop upon the completion of the first non-threading block after the termination of threading mode. The feed hold lamp (SPL lamp), however, lights immediately after the feed hold button (on the machine operator's panel) is pressed. The lamp goes off when feed stops (the CNC enters the single block stop state).

Caution

⚠ CAUTION

1 Feedrate override is ignored during threading, 100% being assumed.
2 During threading, spindle override is ignored, 100% being assumed.
3 When the first non-threading block is executed after threading mode has been finished, and the feed hold button is pressed again (or the feed hold button has been held down), the execution of the non-threading block is stopped immediately.
4 When threading is executed in the single block status, the tool stops after execution of the first block not specifying threading.
5 When the previous block was a threading block, cutting will start immediately without waiting for detection of the 1-turn signal even if the present block is a threading block.
6 When a dry run operation is performed the dry run rate becomes the longitudinal axis feedrate.
7 For T series, the threading retract function is supported only for the threading cycle.

Reference item

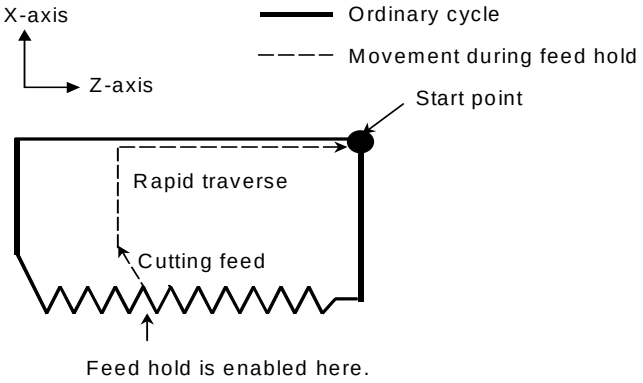
Manual name	Item name
OPERATOR'S MANUAL (T series) (B-64304EN-1)	Constant lead threading
OPERATOR'S MANUAL (M series) (B-64304EN-2)	Threading

6.5.2 Threading Cycle Retract (Canned Cycle) (T Series)

T

Overview

If feed hold is performed during threading (operation 2), the tool immediately retracts while performing chamfering and then returns to the start point in the order of the plane second axis (X-axis) followed by the plane first axis (Z-axis).



The chamfering angle during retraction is the same as that at the end point.

- Thread chamfering

For the thread chamfering angle, 1° to 89° can be specified with parameter No. 5131. When the parameter value is 0, 45° is set.
The type of acceleration/deceleration after interpolation, time constant for acceleration/deceleration after interpolation, and FL feedrate used for thread chamfering are the same as those for threading.

- Retraction after chamfering

For retraction after chamfering, the feedrate, type of acceleration/deceleration after interpolation, and time constant are used as described in the table below.

CFR (bit 0 of parameter No.1611)	Parameter No.1466	Description
0	Non-0	The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No.1626), FL feedrate (parameter No.1627), and feedrate for retraction (parameter No.1466).
0	0	The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No.1626), FL feedrate (parameter No.1627), and rapid traverse rate (parameter No.1420).
1		Before retraction a check is made to see that the specified feedrate has become 0 (delay in acceleration/deceleration is 0), and the type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant and the rapid traverse rate (parameter No. 1420).

For the feedrate for retraction after chamfering, the rapid traverse override can be disabled by setting 1 to bit 4 (ROC) of parameter No.1403.

NOTE
During retraction, the tool does not stop at cutting feedrate override of 0%, regardless of the setting of bit 4 (RF0) of parameter No.1401.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1403	RTV			ROC				

[Input type] Parameter input
[Data type] Bit path

#4 ROC In the threading cycles G92 and G76, rapid traverse override for retraction after threading is finished is:
0: Effective
1: Not effective (Override of 100%)

#7 RTV Rapid traverse override while the tool is retracting in threading
0: Rapid traverse override is effective.
1: Rapid traverse override is not effective.

	Feedrate for retraction in threading cycle G92 or G76
1466	

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/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)
When threading cycle G92 or G76 is specified, retraction is performed after threading. Set a feedrate for this retraction.

NOTE
When this parameter is set to 0 or bit 0 (CFR) of parameter No. 1611 is set to 1, the rapid traverse rate set in parameter No. 1420 is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1610			THLx					

[Input type] Parameter input
[Data type] Bit axis

#5 THLx Acceleration/deceleration in threading cycles
0: Exponential acceleration/deceleration is applied.
1: The same acceleration/deceleration as for cutting feedrate is applied.
(Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610)
As the time constant and FL feedrate, however, the settings of parameter Nos. 1626 and 1627 for threading cycles are used.

	#7	#6	#5	#4	#3	#2	#1	#0
1611								CFR

[Input type] Parameter input
[Data type] Bit path

#0 CFR For retraction after threading in the threading cycles G92 and G76:

0: The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No. 1626) and FL feedrate (parameter No. 1627).

1: The type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant.

NOTE
If this parameter is set to 1, a check is made before a retraction to see that the specified feedrate has become 0 (the delay in acceleration/deceleration has become 0). For retraction, the rapid traverse rate (parameter No. 1420) is used, regardless of the setting of parameter No. 1466. When this parameter is set to 0, parameter No. 1466 is used as the feedrate for retraction. As acceleration/deceleration used for retraction, only acceleration/deceleration after interpolation is used. Rapid traverse before look-ahead interpolation and optimum torque acceleration/deceleration are disabled.

1626	Acceleration/deceleration time constant in threading cycles for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
Set a time constant for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis.

1627	FL rate for acceleration/deceleration in threading cycles 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 an FL feedrate for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis. Set 0 at all times except in a special case.

5131	Cutting angle in thread cutting cycles G92 and G76

[Input type] Parameter input
[Data type] Byte path
[Unit of data] Degree
[Valid data range] 1 to 89
This parameter sets a thread cutting angle in a thread cutting cycle (G92/G76).
When 0 is set, an angle of 45 degrees is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
11223							TRS	

[Input type] Parameter input
[Data type] Bit path

#1 **TRS** In threading cycle retraction, when a block that specifies return to the start point of the threading cycle is executed, threading signal THRD is:
0: Set to 0.
1: Set to 1.

Caution

⚠ CAUTION
Feed hold cannot be performed during retracting.

Note

NOTE
The chamfering angle for retraction is determined by the setting of parameter No. 5131.

Reference item

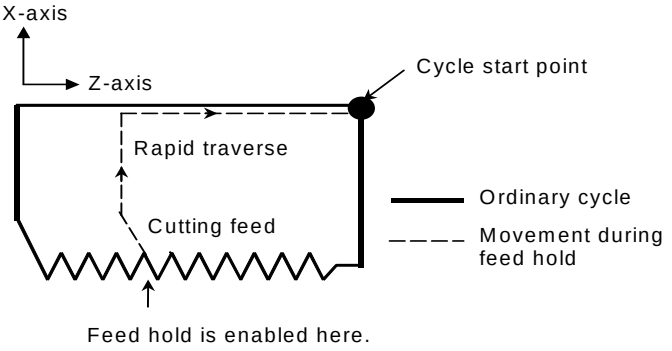
Manual name	Item name
OPERATOR'S MANUAL (Lathe system) (B-64304EN-1)	Threading cycle

6.5.3 Threading Cycle Retract (Multiple Repetitive Canned Cycle) (T Series)

T

Overview

If feed hold is performed during threading in multiple repetitive threading cycle (G76), threading is curtailed (chamfered) and then the tool returns to the start point in the threading cycle and stops.
At this time, if the cycle is started, it resumes from the threading cycle for which feed hold was applied.



The angle of chamfering during retracting is same as that at the end point.

- Thread chamfering

For the thread chamfering angle, 1° to 89° can be specified with parameter No. 5131. When the parameter value is 0, 45° is set.

The type of acceleration/deceleration after interpolation, time constant for acceleration/deceleration after interpolation, and FL feedrate used for thread chamfering are the same as those for threading.

- Retraction after chamfering

For retraction after chamfering, the feedrate, type of acceleration/deceleration after interpolation, and time constant are used as described in the table below.

CFR (bit 0 of parameter No. 1611)	Parameter No. 1466	Description
0	Non-0	The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No.1626), FL feedrate (parameter No.1627), and feedrate for retraction (parameter No.1466).
0	0	The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No.1626), FL feedrate (parameter No.1627), and rapid traverse rate (parameter No.1420).
1		Before retraction a check is made to see that the specified feedrate has become 0 (delay in acceleration/deceleration is 0), and the type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant and the rapid traverse rate (parameter No. 1420).

For the feedrate for retraction after chamfering, the rapid traverse override can be disabled by setting 1 to bit 4 (ROC) of parameter No.1403.

NOTE
During retraction, the tool does not stop at cutting feedrate override of 0%, regardless of the setting of bit 4 (RF0) of parameter No.1401.

⚠ CAUTION
Feed hold cannot be performed during retracting.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1403	RTV			ROC				

[Input type] Parameter input
[Data type] Bit path

#4 ROC In the threading cycles G92 and G76, rapid traverse override for retraction after threading is finished is:
0: Effective
1: Not effective (Override of 100%)

#7 RTV Rapid traverse override while the tool is retracting in threading
0: Rapid traverse override is effective.
1: Rapid traverse override is not effective.

	Feedrate for retraction in threading cycle G92 or G76
1466	

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/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)
When threading cycle G92 or G76 is specified, retraction is performed after threading. Set a feedrate for this retraction.

NOTE
When this parameter is set to 0 or bit 0 (CFR) of parameter No. 1611 is set to 1, the rapid traverse rate set in parameter No. 1420 is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1610			THLx					

[Input type] Parameter input
[Data type] Bit axis

#5 THLx Acceleration/deceleration in threading cycles
0: Exponential acceleration/deceleration is applied.
1: The same acceleration/deceleration as for cutting feedrate is applied.
(Depending on the settings of bits 1 (CTBx) and 0 (CTLx) of parameter No. 1610)
As the time constant and FL feedrate, however, the settings of parameter Nos. 1626 and 1627 for threading cycles are used.

	#7	#6	#5	#4	#3	#2	#1	#0
1611								CFR

[Input type] Parameter input
[Data type] Bit path

#0 CFR For retraction after threading in the threading cycles G92 and G76:
0: The type of acceleration/deceleration after interpolation for threading is used together with the threading time constant (parameter No. 1626) and FL feedrate (parameter No. 1627).
1: The type of acceleration/deceleration after interpolation for rapid traverse is used together with the rapid traverse time constant.

NOTE
If this parameter is set to 1, a check is made before a retraction to see that the specified feedrate has become 0 (the delay in acceleration/deceleration has become 0). For retraction, the rapid traverse rate (parameter No. 1420) is used, regardless of the setting of parameter No. 1466. When this parameter is set to 0, parameter No. 1466 is used as the feedrate for retraction. As acceleration/deceleration used for retraction, only acceleration/deceleration after interpolation is used. Rapid traverse before look-ahead interpolation and optimum torque acceleration/deceleration are disabled.

1626	Acceleration/deceleration time constant in threading cycles for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
Set a time constant for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis.

1627	FL rate for acceleration/deceleration in threading cycles 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 an FL feedrate for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis. Set 0 at all times except in a special case.

5131	Cutting angle in thread cutting cycles G92 and G76

[Input type] Parameter input
[Data type] Byte path
[Unit of data] Degree
[Valid data range] 1 to 89
This parameter sets a thread cutting angle in a thread cutting cycle (G92/G76).
When 0 is set, an angle of 45 degrees is specified.

11223	#7	#6	#5	#4	#3	#2	#1	#0
							TRS	

[Input type] Parameter input
[Data type] Bit path

#1 **TRS** In threading cycle retraction, when a block that specifies return to the start point of the threading cycle is executed, threading signal THRD is:
0: Set to 0.
1: Set to 1.

Caution

⚠ CAUTION
Feed hold cannot be performed during retracting.

Note

NOTE
The chamfering angle for retraction is determined by the setting of parameter No. 5131.

Reference item

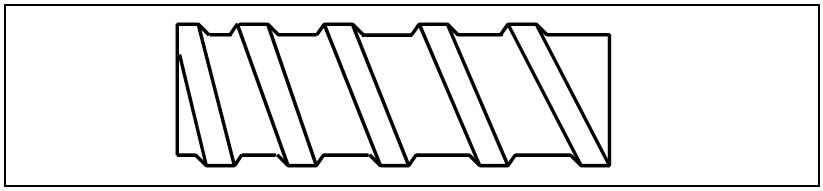
Manual name	Item name
OPERATOR'S MANUAL (Lathe system) (B-64304EN-1)	Multiple repetitive threading cycle

6.5.4 Variable Lead Threading (T Series)

T

Overview

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



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

Alarm and message

Number	Message	Description
PS0313	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.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (Lathe system) (B-64304EN-1)	Variable lead threading

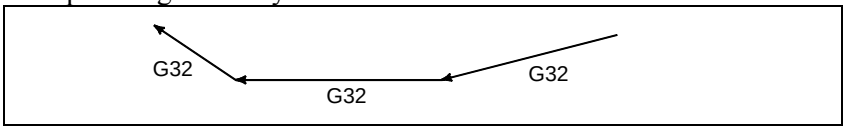
6.5.5 Continuous Threading (T Series)

T

Overview

Threading blocks can be programmed successively to eliminate a discontinuity due to a discontinuous movement in machining by adjacent blocks.

Since the system is controlled in such a manner that the synchronism with the spindle does not deviate in the joint between blocks wherever possible, it is possible to performed special threading operation in which the lead and shape change midway.



Even when the same section is repeated for thread cutting while changing the depth of cut, this system allows a correct machining without impairing the threads.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (Lathe system) (B-64304EN-1)	Continuous treading

6.6 HELICAL INTERPOLATION

Overview

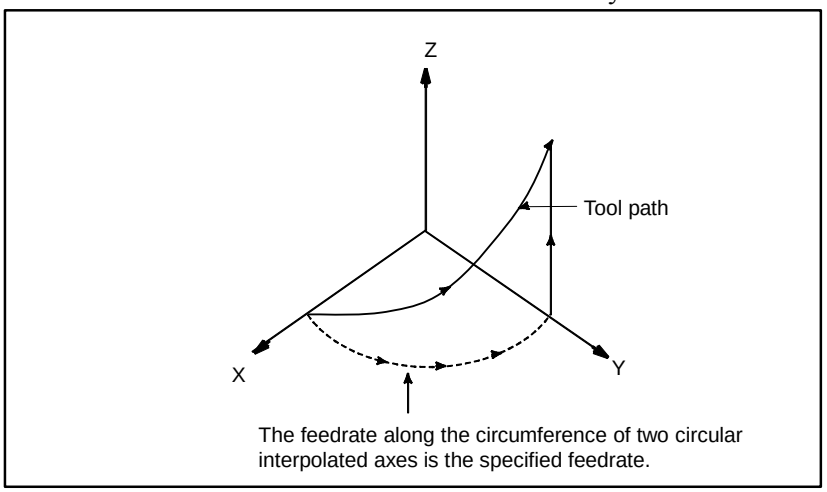
Helical interpolation which moved helically is enabled by specifying up to two other axes which move synchronously with the circular interpolation by circular commands.

NOTE
For Series 0*i*-TD, helical interpolation is an optional function.

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 feed rate along a circular arc, when HTG is specified to 0. Therefore, the feed rate of the linear axis is as follows:

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

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

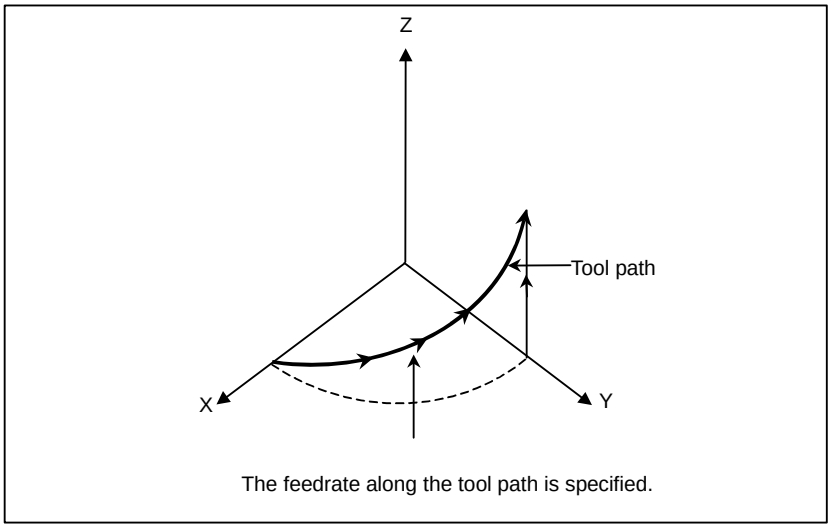


If HTG is set to 1, specify a feedrate along the tool path about the linear axis. Therefore, the tangential velocity of the arc is expressed as follows:

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

The velocity along the linear axis is expressed as follows:

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



Parameter

	#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 is:
0: Specified using the feedrate along the tangent to an arc
1: Specified using the feedrate along axes including a linear axis

Alarm and message

Number	Message	Description
PS0232	TOO MANY HELICAL AXIS COMMAND	Three or more axes were specified as helical axes in the helical interpolation mode.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Helical interpolation

6.7 POLAR COORDINATE INTERPOLATION (T SERIES)

T

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.

Explanation

G12.1 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plane (Fig. 6.7 (a)). Polar coordinate interpolation is performed on this plane.

- Polar coordinate interpolation plane

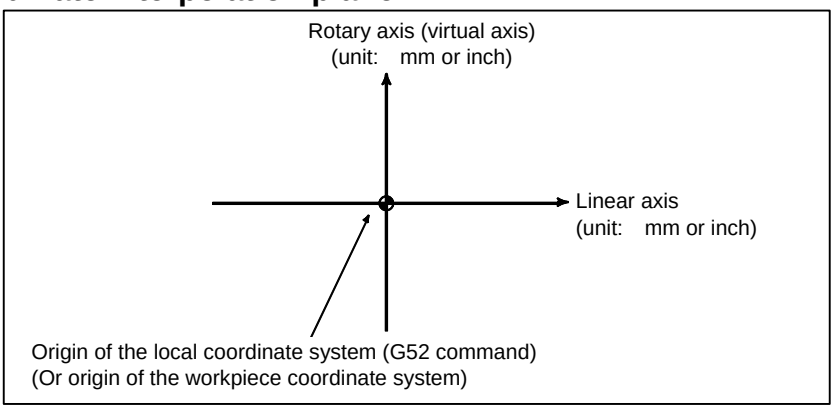


Fig. 6.7 (a) Polar coordinate interpolation plane

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 (No. 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.

Parameter

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.

	#7	#6	#5	#4	#3	#2	#1	#0
5450						PLS		PDI

[Input type] Parameter input
[Data type] Bit path

#0 PDI When the second axis on the plane in the polar coordinate interpolation mode is based on radius specification:
0: Radius specification is used.
1: Diameter specification is used.

#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.

Alarm and message

Number	Message	Description
PS0145	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).

Number	Message	Description
PS0146	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 An illegal G code was specified while in the polar coordinate interpolation mode. The following C codes are not allowed: G27, G28, G30, G31, G37, G52, G92, G53, G17 to G19, G81 to G89, G68 In the 01 group, G codes other than G01, G02, and G03 cannot be specified.
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.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (Lathe System) (B-64304EN-1)	Polar coordinate interpolation

6.8 CYLINDRICAL INTERPOLATION

6.8.1 Cylindrical Interpolation

Overview

The cylindrical interpolation function performs linear interpolation or arc interpolation with another axis by converting the travel distance of a rotation axis specified in degrees to the travel distance on the circumference.

Since programming is allowed with the side of a cylinder developed, a program for cylinder cam grooving or other machining can easily be created.

The rotation axis is regarded as a linear axis and is set in parameter No. 1022 as one of the three basic axes or an axis parallel to them.

For example, when rotation axis C is considered an axis parallel to the X axis, a plane for the Y axis can be selected by specifying G17 and axis addresses C and Y simultaneously (Xp-Yp plane).

Only one rotation axis can be specified for cylindrical interpolation.

Parameter

- **Settings cylindrical interpolation**

	#7	#6	#5	#4	#3	#2	#1	#0
1006							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 rotation axis.
#1 ROSx

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	Rotation 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 parameter No.1008#0(ROAx) and #2(RRLx). (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	Rotation 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 rotation axis roll over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

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), 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	Rotation 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.

1260 **Amount of a shift per one rotation of a rotation 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 amount of a shift per one rotation of a rotation axis.
For the rotation axis used for cylindrical interpolation, set the standard value.

Alarm and message

Number	Message	Description
PS0015	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.
PS0175	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#0 is set to “1” and parameter No. 1260 is set) ON.
PS0176	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.

Reference item

Manual name	Item name
OPERATOR’S MANUAL (B-64304EN)	Cylindrical interpolation

6.9 POLYGON TURNING (T SERIES)

T

Polygon turning means machining a workpiece to a polygonal figure by rotating the workpiece and tool at a certain ratio.

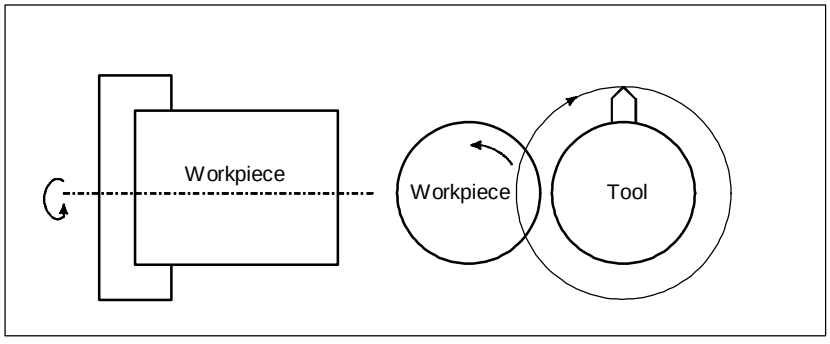


Fig. 6.9 (a) Polygon turning

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 (T series). 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.)

Polygon turning performed using a servo axis as the tool rotary axis is referred to as polygon turning. Polygon turning 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)

When the polygon turning is used, 0 is set in bit 6 (SPG) of parameter No.8133. When the polygon turning with two spindles is used, 1 is set in bit 6 (SPG) of parameter No.8133.
If the polygon turning is performed when 1 is set in SPG, the alarm is issued.

The specifications of polygon turning are described below.
For details about polygon turning with two spindles, see "Polygon Turning with Two Spindles" later in this manual.

6.9.1 Polygon Turning

T

Overview

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 alarm PS0217 to PS0221, PS0314, and PS5018

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.

NOTE
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 Polygon turning with two spindles cannot be used at the same time.
9 G50.2 is the G code for suppressing buffering.

- Spindle connection

A position coder must be mounted on the spindle. However, polygon turning requires no additional changes to the spindle connection.
Polygon turning uses the position coder feedback signal to control the positional relationship (cutting position) between the spindle and tool rotation axis, and the ratio of speed.

- Tool rotation axis (servo axis) connection

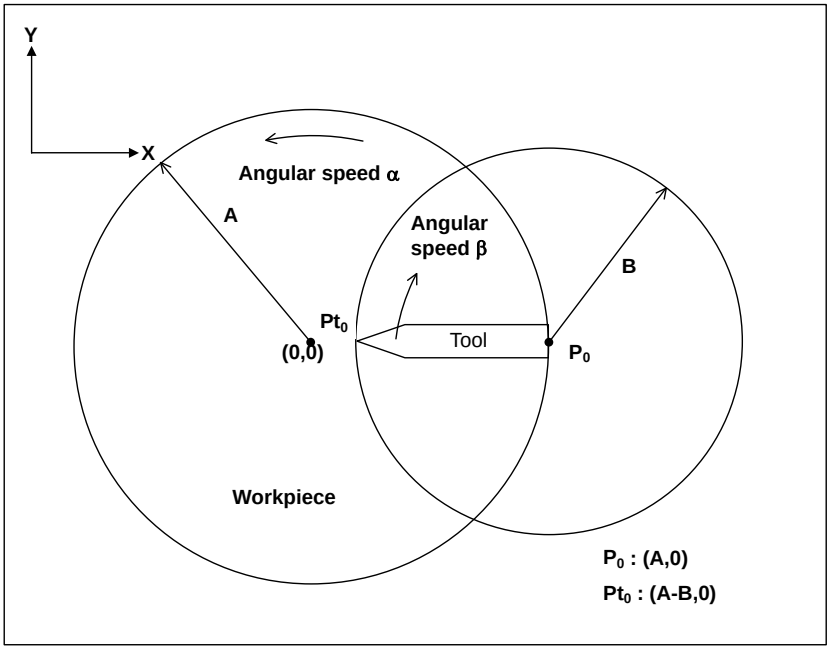
Parameter No. 7610 specifies the controlled axis (servo axis) to be used as the tool rotation axis.
The same parameter setting as for ordinary servo axes applies to the servo axis connection for polygon turning except for some parameters.
When the machine is not in the polygon turning mode, the servo axis specified as the rotation tool axis functions as a feed axis. So, the servo axis can be:

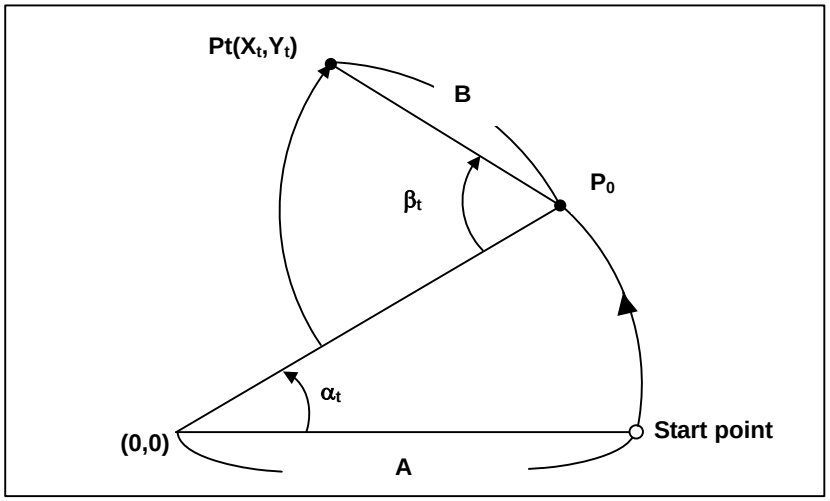
- Used as a subspindle under PMC axis control
- Positioned by a move command from a machining program.

However, be careful about the angle to rotate through and feedrate.
Refer to the concrete example of the following a parameter setting.

- Principle of polygon turning

The principle of polygon turning is explained below. In the figure 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 Po (A, 0) on the workpiece periphery, and the tool nose starts from position Pto (A-B, 0).





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)

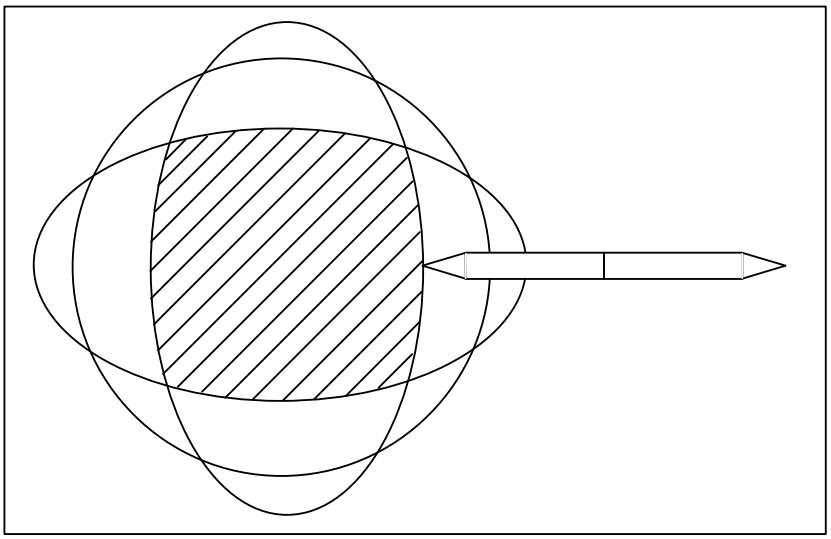
Assuming that the rotation ration of workpiece to tool is 1:2, namely, $\beta=2\alpha$, equations 1 and 2 are modified as follows:

$Xt=A\cos\alpha t-B\cos\alpha t=(A-B)\cos\alpha t$ (Equation 1)'

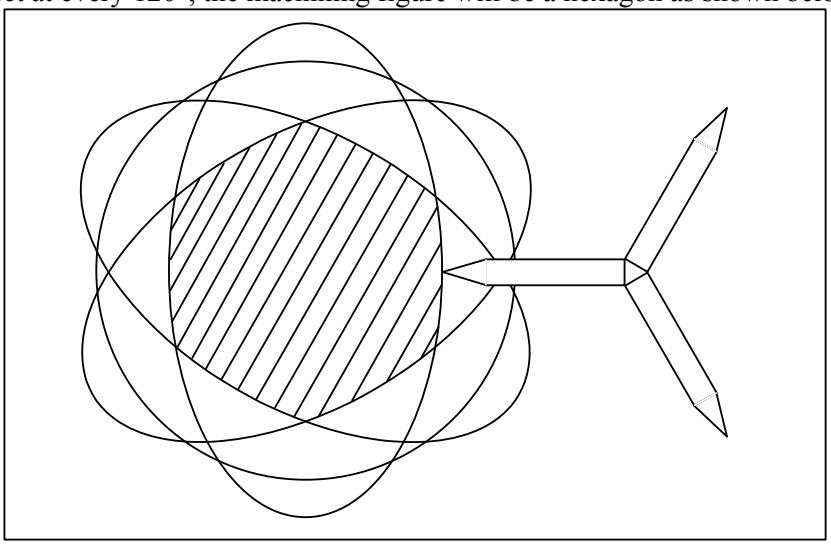
$Yt=A\sin\alpha t+B\sin\alpha t=(A+B)\sin\alpha t$ (Equation 2)'

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.

- Examples of parameter setting

The following descriptions exemplify typical parameter setting for polygon turning using a serial pulse coder (with a million pulse capability).

(1) Tool rotation axis setting
This example uses the CNC's fourth axis (connected as the Y-axis) as a rotation tool axis for polygon turning.
Parameter No. 7610 = 4 (controlled axis number for the tool rotation axis)
The following description assumes that the axis type parameter is set to the fourth axis.

(2) Servo parameter setting
Set the servo parameters as listed below:
CMR=1
DMR=36/100
(With the above setting, the reference counter capacity is 360000.)

No.1820=2 (CMR)
No.1821=360000 (reference counter capacity)
No.2084=36 (DMR numerator)
No.2085=100 (DMR denominator)
For the other servo parameters, specify typical values.

(3) Parameter setting for polygon turning

The least command increment, detection unit, the angle to rotate through per rotation for the Y axis are as follows:

$$\text{Least command increment}=\frac{L\times \text{CMR}}{\text{Q}\times \text{DMR}}$$

$$\text{Detection unit}=\frac{\text{Least command increment}}{\text{CMR}}=\frac{\text{L}}{\text{Q}\times \text{DMR}}$$

$$\text{Angle to rotate through per tool axis rotation}=\frac{360}{\text{Least command increment}}$$

where

L: Tool axis rotation angle per motor rotation (degrees),
(360 × speed increment ratio)

When the servo motor is connected directly to the rotation tool, for example, L = 360. When the tool speed is doubled, L = 720.

Q: Number of pulses per pulse coder rotation
(For a serial pulse coder, Q = 1000000.)

The least command increment specified here is specific to the Y axis. It is determined regardless of what is specified in parameter No. 1013 (ISA/ISC). However, both ISA and ISC must be set to 0 for IS-B setting.

If the servo motor is connected directly to the rotation tool:

$$\text{Least command increment}=\frac{360\times 1}{1000000\times 36/100}=0.001 \quad [\text{deg}]$$

Detection unit=0.001 [deg]

$$\text{Angle to rotate through per tool axis rotation}=\frac{360}{0.001}=360000$$

The upper limit to the tool rotation axis speed is:
Maximum servo motor speed × speed increment ratio
Therefore, if the maximum servo motor speed is 1000 [min^{-1}], and the servo motor is directly connected to the servo motor:
Upper limit to the tool rotation axis speed = 1000 × 1 = 1000 [min^{-1}]
This means the parameters must be set as follows:
No. 7620 = 360000 (angle to rotate through per tool axis rotation)
No. 7621 = 1000 (upper limit to tool rotation axis speed)

(4) Feedrate parameter setting

Because the least command increment is 0.01 [deg], the input unit for the feedrate is 1 [deg/min].
To obtain a rapid traverse speed of 1000 [min^{-1}], for example, specify as follows:
No. 1420 = 360000 (= 1000 × 360)
Other feedrate parameters must be set in degrees per minute.

(5) Commands from the NC program

When the machine is not performing polygon turning, the machining program can issue move commands to the Y-axis.
Such commands can be issued in the same way as for ordinary axes. However, note the travel increment and feedrate.
The Y-axis rotates through 0.003[deg] by the following command:
V3;
Likewise, the polygon axis rotates through 1.000[deg] by the following command:
V1.0;
The current position of the Y-axis in the machine coordinate system is normalized according to the value specified by parameter No. 7620.

Typical values range from 0.000 to 359.999.

Signal

Polygon synchronization under way signal PSYN<Fn063.7>

[Classification] Output signal

[Function] Informs the PMC that the machine is in the polygon turning mode.

[Output cond.] The polygon synchronization signal is set to logical "1" by the polygon turning mode command (G51.2) and stays at "1" during the polygon turning mode.
The signal is reset to logical "0" by the polygon turning mode reset command (G50.2) or a reset. It stays at logical "0" when the machine is not in the polygon turning mode.

NOTE
This signal uses the same address for both polygon turning (using the servo axis) and polygon turning with two spindles.

Other signals

Some signals related to the CNC controlled axis used as the tool rotation axis may be made ineffective depending on whether the machine is in the polygon turning mode.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn063	PSYN							

Diagnosis screen

478	Speed of the tool rotating axis during polygon turning (1/min)

This indication is the speed of the tool rotating axis during polygon turning.

Parameter

	#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.

	#7	#6	#5	#4	#3	#2	#1	#0
7603					PLR			

[Input type] Parameter input

[Data type] Bit path

#3 PLR The machine coordinates of a tool rotation axis for polygon turning are:
0: Rounded by the setting in parameter No. 7620.
1: Rounded by 360° (or the setting in parameter No. 1260 when bit 0 (ROA) of parameter No. 1008 is set to 1).

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).

7620	Movement of tool rotation axis per revolution 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] Real path
[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)
This parameter sets the movement of a tool rotation axis per revolution.

7621	Maximum allowable speed for the tool rotation axis for polygon turning

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min^{-1}
[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 speed of the tool rotation axis.

NOTE
If the speed of a tool rotation axis (polygon synchronization axis) exceeds the set upper limit during polygon machining, clamping is performed at the upper limit. When clamping is performed at the upper limit, the synchronization between the spindle and the tool rotation axis (polygon synchronization axis) deviates. If clamping is performed, an alarm (PS5018) occurs.

Alarm and message

Number	Message	Description
PS0217	DUPLICATE G51.2(COMMANDS)	G51.2 is further commanded in the G51.2 mode. Modify the program.
PS0218	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 display No. 471.
PS0219	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.
PS0220	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.
PS0221	ILLEGAL COMMAND IN SYNCHR-MODE	Polygon turning synchronous operation and Cs contour control or balance cutting are executed at a time. Modify the program.
PS0314	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 with two spindles: Valid spindles are not specified. (Parameter Nos. 7640 to 7643) - A spindle other than the serial spindle. - A spindle is not connected.
PS5018	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 with two spindles: More information as to why this alarm occurred is indicated in diagnosis display No. 471.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (Lathe system) (B-64304EN-1)	Polygon turning

6.9.2 Polygon Turning with Two Spindles

T

Overview

When two or more serial spindles are used, the workpiece rotation axis (master axis) and tool rotation axis (polygon synchronization axis) are synchronized at a certain speed ratio.
With this function, it is also possible to specify the phase difference between the master and polygon synchronization axes.

The polygon turning with two spindles can use different spindle speeds for the same workpiece, because it performs automatic phase compensation when a polygon synchronization mode command is issued or the S command is changed during polygon synchronization mode.

With a two path system, polygon turning is possible on each path.

By default, the first and second spindles in each system are selected as the master axis and polygon synchronization axis for each system. By setting parameters No. 7640 and No. 7641, however, any spindles belonging to the same system can be selected as the master axis and polygon synchronization axis. By setting parameters No. 7642 and No. 7643, however, any spindles belonging to different systems can be selected as the master axis and polygon synchronization axis.

- Command format

The following shows the program command format for polygon turning with two spindles. This is the same as the program command format for polygon turning except for the following points.

1) The command position (R) can be used.
2) Repeated specification in polygon synchronization mode is allowed.

For polygon turning, see also the section about polygon turning.

- Mode command and command value change

The G51.2 command is modal. Once specified, the P, Q, and R values stay unchanged until another G51.2 is issued to change them or polygon synchronized mode is released.

In polygon synchronized mode, the S command for the master axis controls the polygon synchronization axis so that it rotates at a speed of S × Q/P and its phase is R.

- Release command

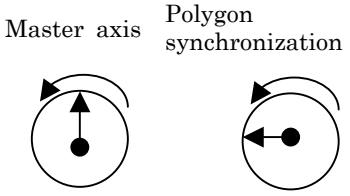
G50.2 command releases the polygon synchronization mode. This mode is released also when:

<1> Reset and emergency stop
(Setting bit 0 (RPL) of parameter No.7603 to 1 prevents polygon synchronization mode from being released.)
<2> Power is turning off.
<3> An alarm condition occurs in the spindle control unit, and the serial spindle control unit stops in an emergency on the PMC signals *ESPA<Gn071.1> and *ESPB<Gn075.1>.
<4> Alarm PS0218, PS0219, PS0221, PS0314, or PS5018 occurs

NOTE
1 G51.2 and G50.2 must be issued separately from other commands.
2 In a G51.2 issued to enter the polygon synchronization mode, R is omissible, but P and Q are required.
3 When G51.2 is specified to change the P, Q, and R settings during the polygon synchronization mode, R can be specified independently. When P and Q are changed, however, be sure to specify both P and Q.
4 When bit 0 (RPL) of parameter No.7603 = 0 regardless of the setting of bit 6 (CLR) of parameter No.3402, a reset issues G50.2 (polygon synchronization release mode). When bit 0 (RPL) of parameter No. 7603 = 1, a reset does not clear G51.2 (polygon synchronization mode) in polygon synchronization mode.

- Spindle operation during the polygon synchronization mode with two spindles

When the start of polygon synchronization is specified by G51.2, the polygon synchronization axis speed is adjusted to Q/P times the master axis speed based on spindle speed command value for the master axis and then phase matching is performed.



M03S100;
When G51.2P1Q1R90.; is specified
Spindle state during polygon synchronization

Master axis speed = S (=100 (min^{-1}))
polygon synchronization axis speed
= S×Q / P
= 100×1 / 1
= 100 (min^{-1})
Phase R = 90 (deg)

When the spindle is accelerated or decelerated or when phase matching is being performed, the synchronization state of rotation ratio of P to Q is not guaranteed. Therefore, in an actual cutting block command, control the spindle speed arrival signal SAR<Gn029.4> after checking the signal state of the polygon spindle speed arrival signal for polygon turning with two spindles PSAR<Fn063.2>=1 or take sufficient wait time in the program.

Each time any of the following changes is made during polygon synchronization, control the speed and then perform phase matching.

1) The spindle speed command for the master axis is changed.
2) The speed ratio commands P and Q are specified by G51.2 again.
3) The phase command value R is specified by G51.2 again.
4) The polygon spindle stop signal *PLSST 0->1 is changed.

How to specify the spindle speed for the master axis (multi-spindle control or spindle output control by PMC) does not change regardless of whether polygon synchronization mode or normal operation is selected.
Spindle speed specification for the polygon synchronization axis is disabled in polygon synchronization mode.

If the speed of the polygon synchronization axis (S × Q / P(min^{-1}): where S is the master axis speed) exceeds the clamp speed set in parameter No. 7621 after the command is executed, an alarm (PS5018) occurs and the polygon synchronization state is cleared.

The diagnostic display bit 2 (QCL) of No. 471 can be used to check whether the clamp speed is exceeded after the command is executed.

- PMC sequence

Although this function is based on the G-code system, it is necessary to add or change PMC ladder sequences because control on the part of the spindle is also required.
(See the description of the PMC sequence in this section.)

- Example of polygon turning with two spindles

This example of polygon turning with two spindles produces a square using single-edged cutting tools (for roughing and finishing).

.		
Mxy; Txxyy;	Step 1.	Mount a roughing tool on the polygon synchronization axis (tool rotation axis).
G00 X100. Z20. M03 S1000 ; . .	Step 2.	Start rotating the workpiece (with the master axis at 1000 (min^{-1}))
G51.2 P1 Q2 R0 ;	Step 3.	Start rotating the tool. In the PMC ladder, the polygon synchronization axis is energized by polygon synchronization under way signal PSYC<Fn063.7>. After acceleration to 2000 (min^{-1}) is performed as the polygon synchronization axis, positioning is performed. The phase is 0. In the PMC ladder, the spindle speed arrival signal SAR<Gn029.4> is controlled based on the polygon spindle speed arrival signal PSAR<Fn063.2> during the polygon synchronization mode.
G01 X80. F10. ; . .	Step 4.	Starts cutting along the X-axis after SAR<Gn029.4> becomes logical 1 in signal control at step 3. Polygonal turning (roughing 1)
G04 P4000 ; .	Step 5.	Retract the tool along the X-axis.
G00 X100. ; .	Step 6.	Change the phase by 180 degrees.
G51.2 R180.; .	Step 7.	Polygonal turning (roughing 2)
[Repeat steps 4, 5, and 6.] .	Step 8.	Release the polygon synchronization mode.
G50.2 ; Myz; Tyyzz; .	Step 9.	Change to a finishing tool. Change the spindle speed for finishing
S2000; G51.2 P1 Q2 R0 ; . .	Step 10.	Start rotating the tool. (master axis at 2000 min^{-1} and polygon synchronization axis at 4000 min^{-1} with a phase difference of 0). Polygonal turning (finishing 1)
[Repeat steps 4, 5, and 6.] .	Step 11.	Change the phase by 180 degrees.
[Repeat step 7.] .	Step 12.	Polygonal turning (finishing 2)
[Repeat steps 4, 5, and 6.] . .	Step 13.	Release the polygon synchronization mode.
G50.2 ; . . .	Step 14.	The polygon synchronization axis (tool rotation axis) stops. The master axis rotates at 2000 (min^{-1}). The master axis stops.
M05S0; . .	Step 15.	

Diagnosis screen

For polygon turning with two spindles, the following information is displayed on the diagnosis display screen.

	#7	#6	#5	#4	#3	#2	#1	#0
470	SC0	LGE		SCF	PSC	PEN	PSU	SPL

Polygon turning with two spindles

Indication of information about the polygon synchronization mode

SPL Polygon synchronization with two spindles under way

PSU Polygon synchronization mode with two spindles being activated

NOTE

If only PSU becomes 1, but no change occurs, and the program stops in a block containing a G51.2 command, the speed of an spindle does not reach the targeted polygon synchronization speed, for example, because bit 7 (PST) of parameter No. 7603 = 0 keeps the spindle from being energized.

PEN Polygon synchronization mode with two spindles released

PSC Spindle speed being changed during polygon synchronization mode with two spindles

SCF Spindle speed changed during polygon synchronization mode with two spindles

LGE The loop gain is different between the spindles during polygon synchronization mode with two spindles.

NOTE

When the speed is changed during polygon synchronization mode, LGE is set to 1 if the spindle synchronization control loop gain used by the serial spindle control unit is different between the master spindle and polygon synchronization axis.

Diagnosis display indicates the loop gain because this function requires that both spindles be controlled with the same loop gain. However, no alarm is issued even if the loop gain is different between the spindles.

(For the serial spindle control unit, the parameters used are changed according to the state of the CTH1 and CTH2 signals.)

SC0 Actual speed command is 0 during polygon synchronization mode with two spindles.

NOTE
Signal SC0 is not a value specified by the program. It is set to 1 under any of the following conditions:
1. When the S command value is adjusted according to the signals related to spindle control, SSTP<Gn029.6> and SOV0-SOV7<Gn030> and the signal related to multi-spindle control <Gn027>, the result is 0.
2. The S command value is smaller than the spindle control resolution (the result of multiplying the S command value by a value of 4095/(maximum spindle speed) is less than 1).
   The S command value is specified by SIND control <Gn032, Gn033>, and it is 0.

If SC0 = 1, the spindle speed becomes 0 and bit 0 of diagnosis display No. 471 becomes 1. In this case, the polygon synchronization rotation ratio is impractical, but alarm PS5018 does not occurs, because it is regarded as the result of the command.

If the following status is indicated during the polygon synchronization mode, there are no abnormalities.

	#7	#6	#5	#4	#3	#2	#1	#0
470	0	0	0	1	-	0	0	1

	#7	#6	#5	#4	#3	#2	#1	#0
471	NPQ	PQE	QMS	NSP	SUO	QCL		SCU

Polygon turning with two spindles
Indication of causes for alarms PS5018, PS0314, and PS0218

#0 to #3 Causes for alarm PS5018
Alarm PS5018 is cleared by a reset, but the indication of its causes remains until the causes are cleared or the polygon synchronization mode is released.

SCU The specified speed is too low during polygon synchronization mode with two spindles. (The unit of speed calculated internally becomes 0.)

NOTE
SCU becomes 1 also when the specified spindle speed is 0 (diagnosis display No. 470#7 = 1). In this case, however, alarm PS5018 is not issued (because the command is 0). When diagnosis display No. 470#7 = 0 and diagnosis display No. 471#0 = 1, alarm PS5018 occurs. Normally this does not occur with speed at which the spindle can rotate.

QCL The polygon synchronization axis is clamped.

NOTE
QCL becomes 1, when the polygon synchronization axis receives a command with a polygon synchronization speed that is higher than the value specified in parameter No. 7621 and is clamped at that speed.

SUO The specified speed is too high during the polygon synchronization mode with two spindles.
(It is clamped to the upper limit calculated internally.)

NOTE
SUO occurs, if a result of (speed specified for the master spindle)/(value specified at P) is higher than 59998. In other words, the master spindle must rotate at a speed lower than 59998 min^{-1} assuming P = 1.

#4 Causes for alarm PS0314
When alarm PS0314 occurs, the polygon synchronization mode is released, but the indication of its causes remains until the alarm PS0314 is cleared by a reset.

NSP A spindle necessary for control is not connected.
(For example, there is not a serial spindle or the second spindle.)
The axis settings for polygon turning are not correct.

#5 to #7 Causes for alarm PS0218
When alarm PS0218 occurs, the polygon synchronization mode is released, but the indication of its causes remains until the alarm PS0218 is cleared by a reset.

QMS When bit 1 (QDR) of parameter No. 7603 = 1, a negative value is specified at Q.

PQE In a G51.2, either P or Q has a value out of the specifiable range.
Or, P and Q are not specified as a pair.

NPQ In a G51.2, R is specified when P and Q have not been specified at all, or none of P, Q, and R has been specified.

Indication of values specified during the polygon synchronization mode with two spindles

474	**Rotation ratio for the master axis during the polygon synchronization mode with two spindles (P command value)**

This indication is the current rotation ratio (P command value) of the master axis during the polygon synchronization mode with two spindles.

475	**Rotation ratio for the polygon synchronization axis during the polygon synchronization mode with two spindles(Q command value)**

This indication is the current rotation ratio (Q command value) of the polygon synchronization axis during the polygon synchronization mode with two spindles.

476	**Phase difference between the two spindles under polygon synchronization control with two spindles(R command value)**

This indication is the current phase value (R command value) specified during the polygon synchronization mode with two spindles. (The increment system for the reference axis used.)

However, if bit 5 (RDG) of parameter No. 7603 = 1, the indication is the amount of shifting specified for the serial spindle (number of pulses after conversion is performed assuming 360 degrees = 4096 pulses).

Indication of the actual speed of each spindle during the polygon synchronization mode with two spindles

477	Actual master axis speed (min^{-1}) during the polygon synchronization mode with two spindles

This indication is the actual speed of the master axis during the polygon synchronization mode with two spindles.

478	Actual polygon synchronization axis speed (min-1) during the polygon synchronization mode with two spindles

This indication is the actual speed of the polygon synchronization axis during the polygon synchronization mode with two spindles.

Spindle screen

During the polygon synchronization mode with two spindles, information for the spindle synchronization control mode is displayed.

The following information is displayed on the spindle screen.
"Positional deviation" indicates the positional deviation of the spindle currently being displayed.
"Synchronization error" indicates the absolute value of a synchronization error in polygon synchronization during polygon turning with two spindles. The synchronization error value is displayed on the spindle screen for the spindle used as the polygon synchronization axis.

Actual spindle speed (1/MIN)
Motor speed (1/MIN)
Actual loop gain (0.01/S)
Spindle positional deviation (PLS)
Synchronization error (PLS)

Signal

Polygon spindle stop signal *PLSST<Gn038.0>

[Classification] Input signal

[Function] This function is enabled when bit 7 (PST) of parameter No. 7603 = 1. This signal is used to stop the spindle during the polygon synchronization mode with two spindles.
"0" = polygon spindle stop
"1" = polygon spindle operable
During the polygon synchronous mode with two spindles, the rotation of the spindle is controlled with a position loop created.
The spindle is rotated by move command pulses based on the polygon synchronization speed.
When the activated spindle is turned off by the spindle stop (for example, M05) command, move command pulses are generated even when activation is off unless the spindle speed command is set to 0 using the spindle stop signal *SSTP<Gn029.6> or the like, some error pulses being accumulated. When activation is turned on at this time, the spindle may rotates at high speed because the accumulated error pulses are moved at a time.
This signal can be used to stop the delivery to the polygon spindle only during the polygon synchronous mode with two spindles. The signal must be controlled according to the activation status (on/off) of the polygon spindle.

For details on usage, see the section that describes PMC sequences.

Polygon synchronization under way signal PSYN<Fn063.7>

[Classification] Output signal

[Function] Informs the PMC that the system is in the polygon synchronization mode.

[Output cond.] The polygon synchronization mode command (G51.2) sets this signal to logical "1". It stays at "1" as long as the system is in the polygon synchronization mode. It is turned to "0" when the polygon synchronization mode is cleared (G50.2 command or a reset). It stays at "0" when the system is not in the polygon synchronization mode.

NOTE
The same address is used for this signal in both polygon turning (using the servo axis) and the polygon turning with two spindles.

Polygon spindle speed arrival signal PSAR<Fn063.2>

[Classification] Output signal

[Function] Informs the PMC that the spindle has reached its constant-speed for polygon synchronization during polygon turning with two spindles.

[Output cond.] During polygon synchronization mode with two spindles, whether the constant-speed is reached for polygon synchronization is output as shown below:
"0" = not reached (during phase change or acceleration/deceleration under way)
"1" = reached
During the polygon synchronization mode, this signal becomes logical "1" when the speed of each spindle reaches the acceptable level specified in parameter No. 7631 and remains there for a period specified in parameter No. 7632.
If the speed of either spindle goes off the acceptable level, or a change is made to the S command, the signal returns to logical "0" and begins monitoring the above condition.
When this signal is "0", the specified speed ratio and phase are not guaranteed for polygon turning. If the signal is confirmed before actual turning is started, however, the operation is more efficient than when a dwell command (like G04) is used to allow wait time.

Polygon master axis not arrival signal PSE1<Fn063.0>
Polygon synchronization axis not arrival signal PSE2<Fn063.1>

[Classification] Output signal

[Function] Informs the PMC whether the actual speed of each spindle has reached the specified speed during polygon turning mode with two spindles.

[Output cond.] During the polygon synchronization mode with two spindles, whether each spindle has reached the polygon synchronization speed is output as shown below:
"0" = reached
"1" = not reached (during phase change or acceleration/deceleration under way)
During the polygon synchronization mode with two spindles, this signal becomes logical "1" when the speed of master axis and polygon synchronization axis does not reach the acceptable level specified in parameter No. 7631.

Signal address

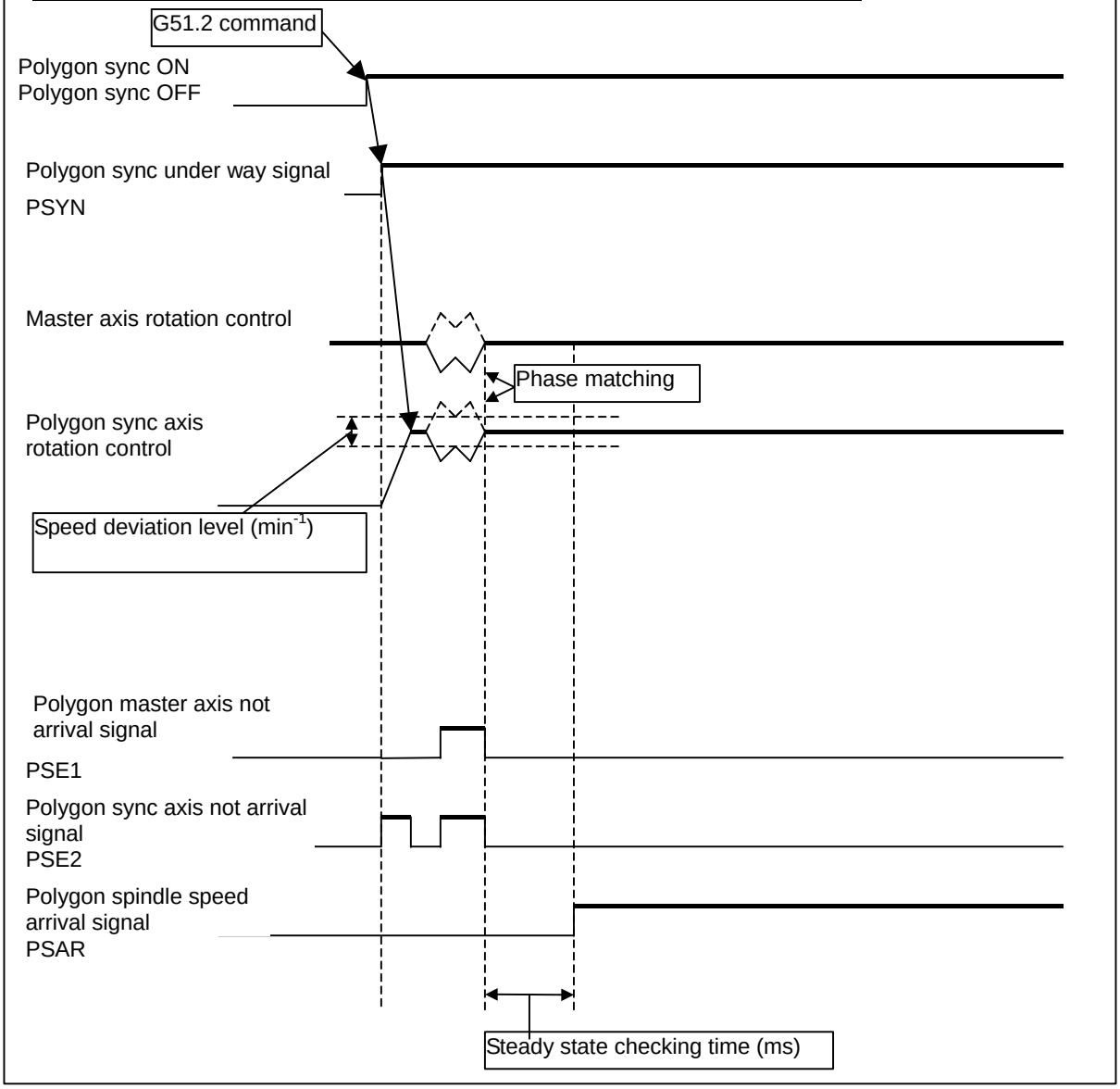
	#7	#6	#5	#4	#3	#2	#1	#0
Gn038								*PLSST

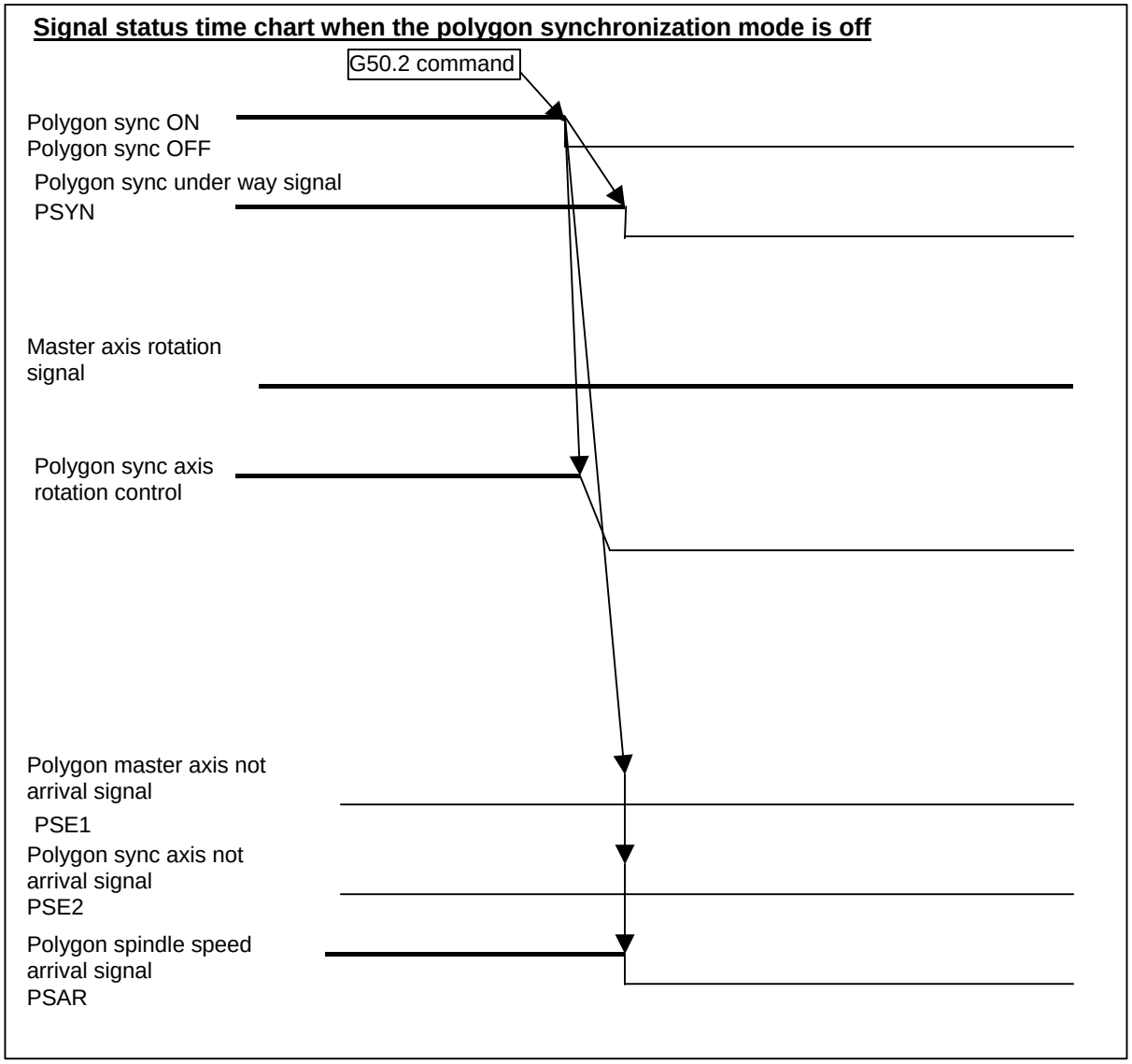
	#7	#6	#5	#4	#3	#2	#1	#0
Fn063	PSYN					PSAR	PSE2	PSE1

PMC sequence

The following shows the signal status time chart when the polygon synchronization mode is on and off.

Signal status time chart when the polygon synchronization mode is on





When the polygon synchronization mode is started by the G51.2 command, the polygon synchronization under way signal PSYN<Fn063.7> is set to 1.
Create a PMC sequence during the polygon synchronization mode by checking the polygon synchronization under way signal with the PMC ladder.

First, select one method of controlling spindle activation used during the polygon synchronization mode from methods A and B. Then create a PMC sequence according to the selected method.

- Method A

During the polygon synchronization mode, activation of the master axis and polygon synchronization axis is automatically turned on.
During this mode, activation of the spindle is not turned off.

For method A, set bit 7 (PST) of parameter No. 7603 to 0.
In a PMC sequence, activation of the master axis and polygon synchronization axis must be turned on when the polygon synchronization under way signal PSYN<Fn063.7> changes from 0 to 1.

In addition, while the polygon synchronization under way signal is 1, prevent the activation of the master axis and polygon synchronization axis from being turned off by, for example, prohibiting the acceptance of spindle stop (for example M05) during normal control.

when the polygon synchronization under way signal PSYN<Fn063.7> changes from 1 to 0, basically turn off activation.

The NC stops at the G51.2 command block for entering the polygon synchronization mode and does not proceed to the next block until the spindle speed reaches the polygon synchronization speed.

- Method B

Even during the polygon synchronization mode, another M code or the like is used to turn on or off activation of the spindle.
Or, even during the polygon synchronization mode, activation of the spindle may be turned off.

For method B, set bit 7 of parameter No. 7603 (PST) to 1.
This parameter setting enables the polygon spindle stop signal *PLSST<Gn038.0> and makes the NC proceed to the next block without waiting for the spindle speed to reach the polygon synchronization speed in the G51.2 command block for starting the polygon synchronization mode.

In a PMC sequence, during the polygon synchronization mode (the polygon synchronization under way signal PSYN<Fn063.7> is 1), set the polygon spindle stop signal *PLSST<Gn038.0> to 1 after confirming that activation of the master axis and polygon synchronization axis is turned on.
When the polygon synchronization mode is not entered (the polygon synchronization under way signal PSYN<Fn063.7> is 0), set the polygon spindle stop signal *PLSST<Gn038.0> to 0.

When the polygon spindle stop signal *PLSST<Gn038.0> is changed from 0 to 1 during the polygon synchronization mode, phase control is performed after acceleration from the spindle stop state to the polygon synchronization speed is performed.

- Sequences common to methods A and B

During the polygon synchronization mode, observe the follow rules to create a PMC sequence regardless of whether method A or method B is used.

(1) The rotation direction of the master axis during the polygon synchronization mode is fixed to the forward direction rather than being switched by the forward/reverse rotation specification signal (SFR/SRV). When changing the rotation direction, use a command.
- When using the S command in the program
  Specify M03/M04 by setting bit 7 of parameter No. 3706 to 1.
- When using a command by the spindle output control by the PMC
  Change the rotation direction by setting the spindle motor command polarity selection signal SSIN<Gn033.6> to "1" to control the spindle motor command polarity specification signal SGN<Gn033.5>.
  The rotation direction of the polygon synchronization axis must also be fixed to the forward direction rather than being switched by the SFR/SRV signal.

(2) During the polygon synchronization mode, phase control is performed after the polygon synchronization speed is reached. Therefore, the speed arrival signal of each spindle (SARA<Fn045.3> and SARB<Fn049.3>) is insufficient for determining the start condition of the cutting feed command.
To confirm that the spindle speed is reached during the polygon synchronization mode, use the polygon spindle speed arrival signal PSAR<Fn063.2> to control the spindle speed arrival signal SAR<Gn029.4>.
The polygon spindle speed arrival signal PSAR<Fn063.2> can be used to check whether both spindles meet the settings of parameters No. 7631 and 7632 after phase control is performed in the polygon synchronization mode.

The set time (parameter No. 3740) that elapses before the spindle speed arrival signal is checked is also valid for the execution of the cutting feed command during the polygon synchronization mode.

If the spindle speed arrival signal SAR<Gn029.4> is not used as the start condition for the cutting feed command, the time required for the master axis and polygon synchronization axis to rotate at steady-state speed must be reserved during start of the polygon synchronization mode, change of the spindle speed, or change of the machining condition by a program command (for example, G04) before cutting by polygon turning is started.

(3) During the spindle polygon synchronization mode, the polygon synchronization axis cannot be rotated independently of the master axis.
Since the spindle orientation function (ORCMA <Gn070.6> and ORCMB<Gn074.6>) cannot also be used, the polygon synchronization mode must be released before shifting gears or changing tools or workpieces.
If necessary, take such an action that displays a message requesting the release of the polygon synchronization mode when an unavailable command is used during the polygon synchronization mode.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7603	PST		RDG				QDR	RPL

[Input type] Parameter input
[Data type] Bit path

#0 RPL Upon reset, polygon turning mode with two spindles is:
0: Released.
1: Not released.

#1 QDR The rotational direction of the polygon synchronization axis:
0: Depends on the sign (+/-) of a specified value for Q.
1: Depends on the rotational direction of the first spindle.
If a negative value is specified for Q when QDR = 1, the alarm (PS0218) is issued.

#5 RDG On the diagnosis screen No.476, for spindle-spindle polygon phase command value (R), displays:
0: The specified value (in the increment system for the rotation axis).
1: The actual number of shift pulses.

NOTE
A phase command is specified in address R, in units of degrees. For control, the actual shift amount is converted to a number of pulses according to the conversion formula: 360 degrees = 4096 pulses. This parameter switches the display of a specified value to that of a converted value.

#7 PST The polygon spindle stop signal *PLSST <Gn038.0>is:
0: Not used.
1: Used.

7621	Maximum allowable speed for the tool rotation axis for polygon turning

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min^{-1}
[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 speed of the tool rotation axis.

NOTE
If the speed of a tool rotation axis (polygon synchronization axis) exceeds the set upper limit during polygon machining, clamping is performed at the upper limit. When clamping is performed at the upper limit, the synchronization between the spindle and the tool rotation axis (polygon synchronization axis) deviates. If clamping is performed, an alarm (PS5018) occurs.

7631	Allowable spindle speed deviation level in polygon turning with two spindles

[Input type] Parameter input
[Data type] Word path
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999
This parameter sets the allowable level of deviation between the actual speed and specified speed of each spindle in polygon turning with two spindles. The value set with this parameter is used for both the master axis and polygon synchronization axis.
When 0 is set in this parameter, the specification of 8 [min^{-1}] is assumed.

7632	Steady state confirmation time duration in spindle polygon turning

[Input type] Parameter input
[Data type] Word path
[Unit of data] msec
[Valid data range] 0 to 32767
This parameter sets the duration required to confirm that both spindles have reached their specified speeds in polygon turning with two spindles.
If the state where the speed of each spindle is within the range set with parameter No.7631, and has lasted at least for the duration specified with parameter No.7632, the spindle polygon speed arrival signal PSAR <Fn063.2> is set to 1.
When 0 is set in this parameter, the specification of 64 [msec] is assumed.

7640	Master axis in polygon turning with two spindles

[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 polygon turning with two spindles.

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 an axis other than the first serial spindle is selected as the master axis, multi-spindle control is required to execute 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 polygon turning with two spindles

[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 polygon turning with two spindles.

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 an axis other than the first serial spindle is selected as the master axis, multi-spindle control is required to execute 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 polygon turning with two spindles (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 polygon turning with two spindles.

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 an axis other than the first serial spindle is selected as the master axis, multi-spindle control is required to execute 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 polygon turning with two spindles (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 polygon turning with two spindles.

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 an axis other than the first serial spindle is selected as the master axis, multi-spindle control is required to execute 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.

Alarm and message

Number	Message	Description
PS0194	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 simple spindle synchronous control mode (M series).

Number	Message	Description
PS0218	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 display No. 471.
PS0219	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.
PS0221	ILLEGAL COMMAND IN SYNCHR-MODE	Polygon turning synchronous operation and Cs contour control or balance cutting (T series (2-path)) are executed at a time. Modify the program.
PS0314	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 with two spindles: Valid spindles are not specified. (Parameter Nos. 7640 to 7643) - A spindle other than the serial spindle. - A spindle is not connected.
PS5018	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 with two spindles: More information as to why this alarm occurred is indicated in diagnosis display No. 471.

Caution

⚠ CAUTION

1 The maximum spindle speed for each gear stage (No. 3741 to 3744) must be specified correctly according to the model of the machine. In addition, ordinary spindle connections must have been terminated.

2 This function uses the one-rotation signal for the spindle as a reference point for phase adjustment.
When a built-in sensor is used, and there are gears between the spindle and spindle motor, it is necessary to install a detector on the spindle separately to take a one-rotation signal from the spindle. If the detector does not guarantee detection of a correct position from arbitrary speed, set bit 4 (HST) of parameter No. 7602 and bit 7 (RFCHK3) of serial spindle parameter No. 4016 to 1. This setting reduces the spindle speed automatically down to 0 for spindle position detection each time the polygon turning mode with two spindles is entered, thus guaranteeing a correct phase relationship during polygon synchronization mode with two spindles.

3 This function uses the spindle synchronization function for serial spindles. (However, it does not require the spindle synchronization option for the CNC.) You may need to specify the relevant serial spindle parameters (such as Nos. 4032 to 4035 and Nos. 4065 to 4068).
Specify the same serial spindle loop gain (parameters Nos. 4065 to 4068) for both spindles.
If the same serial spindle loop gain is not used for both spindles, polygon turning may not be accurate.
If an attempt is made to perform polygon turning with two spindles using different loop gain, diagnosis display LGE (No. 470#6) becomes 1. (No alarm is issued.)

⚠ CAUTION

4 Before using the polygon synchronization mode, place both master axis and polygon synchronization axis in the spindle control mode. If the polygon synchronization mode starts during another mode (Cs contour control mode, spindle orientation, or the like), an alarm occurs.

5 During the polygon synchronization mode, no command cannot be executed for the polygon synchronization axis.
During the polygon synchronization mode, the spindle orientation function cannot be used for either the master axis and polygon synchronization axis. Therefore, gear, tool or workpiece change is basically unusable during the polygon synchronization mode.

6 During polygon synchronization mode, speed change and phase adjustment are performed each time the spindle speed is changed. Therefore, this mode cannot be used together with a function that causes continuous spindle speed change (such as G96 constant surface speed control)

7 During the polygon synchronization mode, the rotation ratio between the master axis and polygon synchronization axis is controlled with priority. Therefore, the difference between the master axis speed and S command value may become larger than during ordinary spindle control.

Note

NOTE

1 During the polygon synchronization mode, phase control is performed in the least command increment of 360/4096 = 0.08789...(degrees) in reference to the one-rotation signal for each spindle. A setting less that the least command increment is actually invalid.
Since phase control covers relative phase of both spindles within a turn, a setting greater than 360 degrees has no meaning.
This function does not limit the range of valid R settings. A setting greater than 360 degrees is assumed as the remainder of division by 360.

2 This function cannot be used with polygon turning.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (Lathe system) (B-64304EN-1)	Polygon turning

6.10 NORMAL DIRECTION CONTROL (M SERIES)

M

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. 6.10 (a) Sample Movement of the tool).

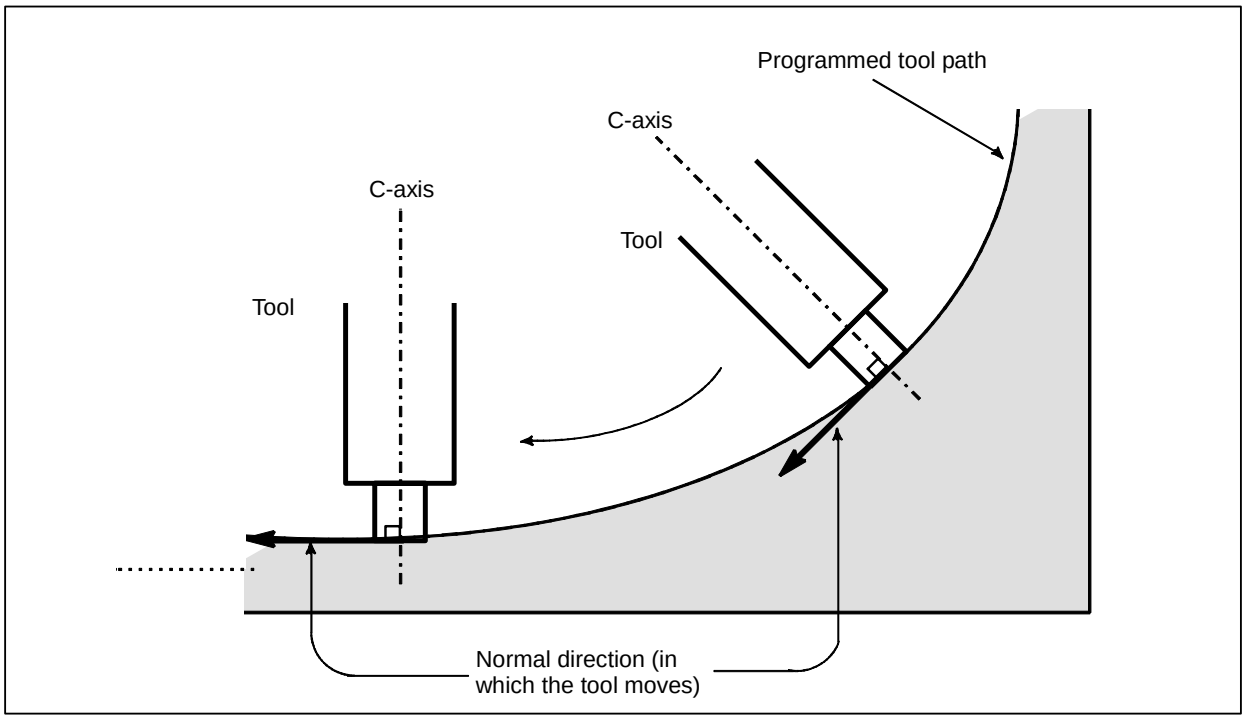


Fig. 6.10 (a) Sample Movement of the tool

Movement of the tool inserted at the beginning of each block is executed at the feedrate set in parameter No. 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 rate is applied.
If the feedrate of the C axis exceeds the maximum cutting feedrate 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 feedrate of the C axis.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1006							ROSx	ROTx

[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 ROTx Setting linear or rotation axis.

#1 ROSx

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	Rotation 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 parameter No.1008#0(ROAx) and #2(RRLx). (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	Rotation 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 rotation axis roll over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

NOTE
Just a rotation axis can be subjected to normal direction control.

5480	
	Number of the axis for controlling the normal direction

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to the maximum controlled axis number
This parameter sets the controlled axis number of the axis for controlling the normal direction.

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.

5482	
	Limit value used to determine whether to ignore the rotation insertion of the normal direction controlled axis

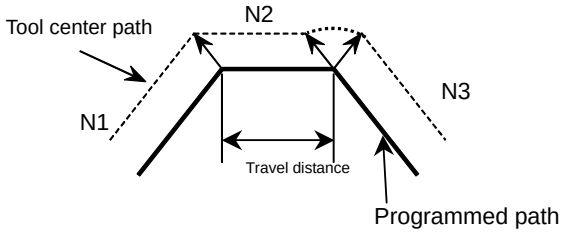
[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
The rotation block of the normal direction controlled axis is not inserted when the rotation insertion angle calculated during normal direction control does not exceed this setting.

The ignored rotation angle is added to the next rotation insertion angle, and the block insertion is then judged.

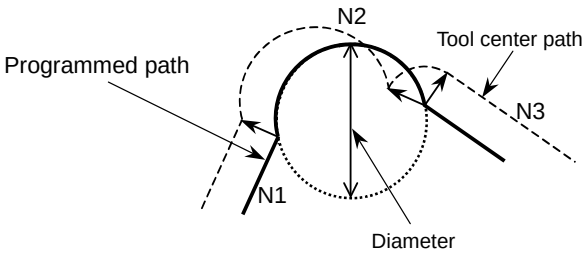
NOTE
1 No rotation block is inserted when 360 or more degrees are set.
2 If 180 or more degrees are set, a rotation block is inserted only when the circular interpolation setting is 180 or more degrees.

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)



For straight line
When the travel distance of N2 in the figure on the left does not exceed the setting, block N2 is machined with the tool being normal to block N1.



For arc
When the arc diameter of N2 in the figure on the left does not exceed the setting, arc N2 is machined with the tool being normal to block N1. A normal direction axis is not controlled to move in the normal direction according to the arc movement.

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.
The cutting feed rate is clamped so that the feedrate in the tangent direction does not exceed the parameter setting.

6.11 LINEAR INTERPOLATION (G28, G30, G53)

Overview

When positioning operation of linear interpolation type is specified (parameter LRP (No. 1401#1) = 1), the following operations can also be set as operations of linear interpolation type by setting parameter ZRL (No. 1015#4) to 1:

- Movement from an intermediate point to a reference position in automatic reference position return operation (G28)
- Movement from an intermediate point to a reference position in second, third, or fourth reference position return operation (G30)
- Positioning by machine coordinate system selection (G53)

When any of the operations above is set as an operation of linear interpolation type, acceleration/deceleration control follows the setting of parameter PRT (No. 1603#4).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1015				ZRL				

[Input type] Parameter input
[Data type] Bit path

#4 ZRL When a reference position is established, the tool path from the intermediate point to the reference position in automatic reference position return (G28) and machine coordinate positioning (G53) are based on:
0: Positioning of nonlinear interpolation type
1: Positioning of linear interpolation type

NOTE
This parameter is valid when bit 1 (LRP) of parameter No. 1401 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1603				PRT				

[Input type] Parameter input
[Data type] Bit path

#4 PRT For positioning of linear interpolation type:
0: Acceleration/deceleration of acceleration fixed type is used.
1: Acceleration/deceleration of time fixed type is used.

Note

NOTE

Automatic reference position return operation of low-speed type (G28)
If reference position return operation is not performed for a specified axis even once after the power is turned on in automatic reference position return operation (G28), a reference position return operation of low-speed type is performed for the axis.

At this time, positioning of non-linear interpolation type is performed.
For example, if G28X0Y0Z0; is specified when reference position return operation is completed for the X-axis and Y-axis, and is not completed for the Z-axis, positioning of non-linear interpolation type is performed.

6.12 NANO SMOOTHING (M SERIES)

M

Overview

When a desired sculptured surface is approximated by minute segments, Nano smoothing function generates a smooth curve inferred from the programmed segments and performs necessary interpolation.
Nano smoothing function infers a curve from a programmed figure approximated with segments within tolerance. The interpolation of the curve reduces the segment approximation error, and the nano interpolation makes the cutting surface smoother.
For this function, the AI contour control II option is required.

Format

G5.1 Q3 Xp0 Yp0 Zp0 ; : 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

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 Nano smoothing mode on block.
The specified axis is subjected to Nano smoothing, but no movement is made even in the absolute programming mode.
3. Nano smoothing mode is also canceled by 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 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

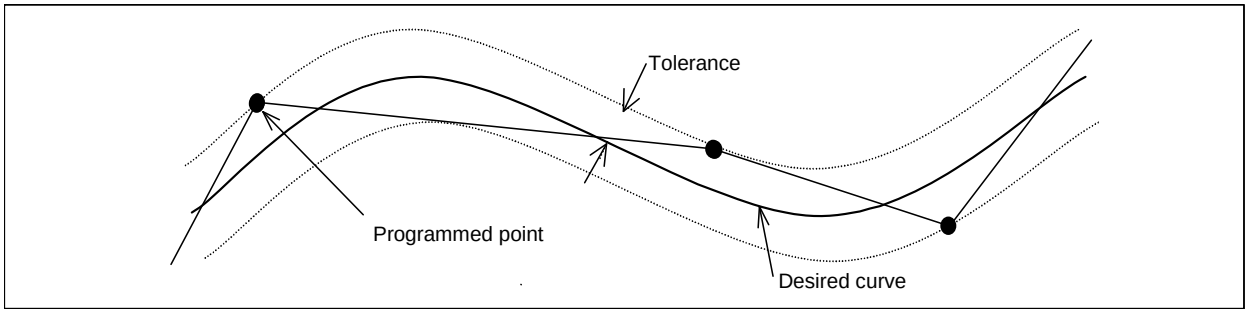
If specifying the machining condition selecting function, specify G5.1 Q1 Rx first and then Nano smoothing.

Example

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

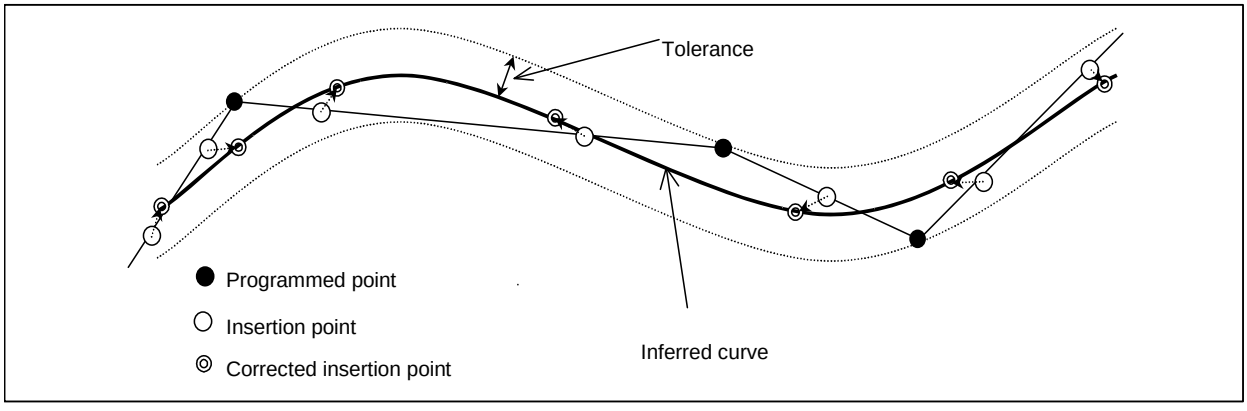
Explanation

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



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. 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.



- Specifying the tolerance

The tolerance of the program of Nano smoothing is specified in parameter No. 19581.

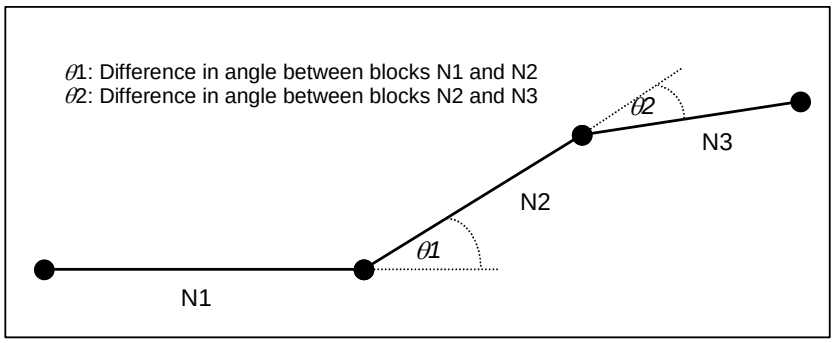
The insertion points are corrected within tolerance, and a curve is inferred accordingly.
If 0 is specified in parameter No. 19581, the minimum travel distance in the increment system is considered to be the tolerance.

- 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 Nano smoothing mode, 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 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.

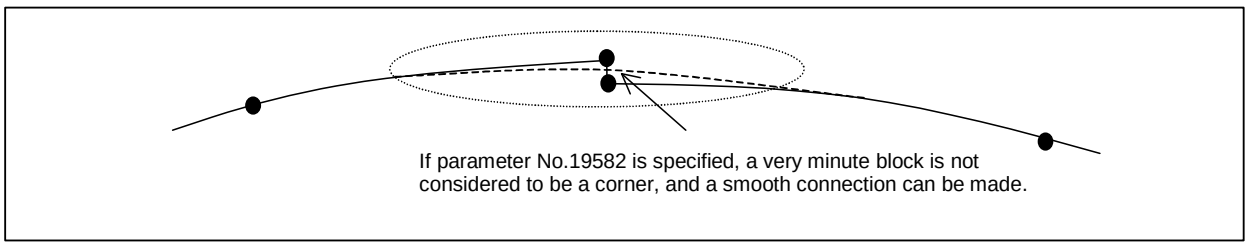
- Making a decision at a corner

If the difference in angle between adjacent programmed blocks exceeds the value specified in parameter No. 8487 in Nano smoothing mode, 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 Nano smoothing mode is canceled in a block, Nano smoothing for the rotation axes is not performed, either.



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.



- Automatically turning on and off AI contour control II with Nano smoothing

Specifying G5.1 Q3 also enables Nano smoothing and AI contour control II to be turned on at the same time.
The automatic velocity control by AI contour control II reduces impacts on the mechanical system.
Specifying G5.1 Q0 cancels Nano smoothing and the AI contour control II 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
- Tool radius compensation cancel
- Canned cycle cancel
- Scaling cancel
- Macro modal call cancel
- Constant surface speed control cancel
- Cutting mode
- Coordinate system rotation cancel
- Polar coordinate command cancel
- Normal direction control cancel
- Programmable mirror image 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 Nano smoothing

Diagnostic data (No. 5000) indicates whether Nano smoothing mode is enabled in the current block.
If 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.

G15 : Polar coordinate command cancel
G40 : Tool radius compensation cancel
G40.1 : Normal direction control cancel
G49,G43,G44 : Tool length compensation cancel or tool length compensation
G50 : Scaling cancel
G50.1 : Programmable mirror image cancel
G64 : Cutting mode cancel
G67 : Macro modal call cancel
G69 : Coordinate system rotation cancel

G80 : Canned cycle cancel
G94 : Feed per minute
G97 : Constant surface speed control cancel

- Single block operation

When single block operation is carried out in Nano smoothing mode, the operation stops at a corrected insertion point not at a programmed point.
Even in 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 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.

- Cutter compensation

If cutter compensation is specified in Nano smoothing mode, Nano smoothing mode is cancelled.
Then, when the command of cutter 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 cutter compensation specified in Nano smoothing mode, irrespective of the parameter setting.
A command related to cutter compensation should not be specified in Nano smoothing mode unless it is absolutely necessary.

- Interrupt-type custom macro

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

- Manual intervention

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

- Number of blocks that can be specified successively

Up to about 300,000,000 blocks can be specified successively in 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 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 Nano smoothing mode.
Therefore, the programmed commands must be executed continuously in 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 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 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) Retracing (Retrace)
Retracing cannot be performed in Nano smoothing mode.

(3) Manual handle retrace
In Nano smoothing mode, manual handle retrace cannot be performed.

- Dynamic graphic display

The dynamic graphic display function draws the path in Nano smoothing mode by linear interpolation.

Parameter

8486	
	Maximum travel distance of a block where 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 Nano smoothing. If the line specified in a block is longer than the value set in the parameter, Nano smoothing is not applied to that block.

8487	
	Angle at which 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 Nano smoothing.
At a point having a difference in angle greater than this setting, Nano smoothing is turned off.

8490	
	Minimum travel distance of a block where 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 Nano smoothing.
If the line specified in a block is shorter than the value set in this parameter, Nano smoothing is not applied to that block.

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.

Alarm and message

Number	Message	Contents
PS0340	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 intervention.
PS0341	TOO MANY COMMAND BLOCK (NANO SMOOTHING)	There are more blocks than can be commanded consecutively in Nano smoothing mode.
PS0342	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.
PS0343	ILLEGAL COMMAND IN NANO SMOOTHING	G43, G44, or G49 was commanded during a Nano smoothing.
PS0344	CANNOT CONTINUE NANO SMOOTHING	An illegal command or operation by which a Nano smoothing could not be continued was performed.

Reference item

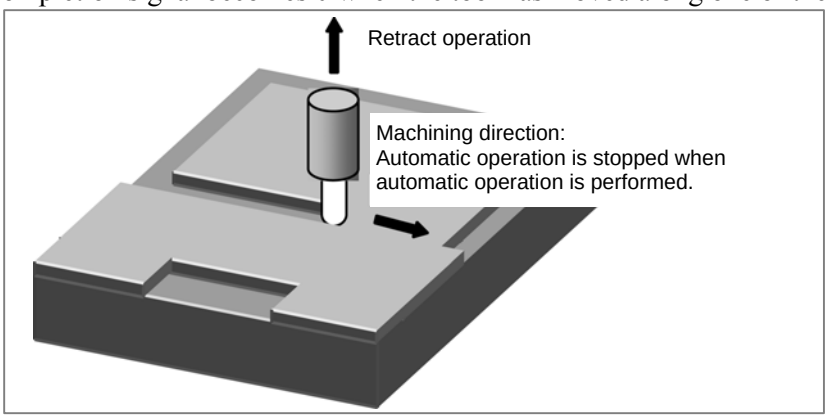
Manual name	Item name
For Machining Center System OPERATOR'S MANUAL (B-64304EN-2)	3.3 Nano smoothing

6.13 GENERAL PURPOSE RETRACT

General

In automatic operation mode or in manual operation mode, setting the retract signal RTRCT to 1 causes this function to capture the rise of this signal, causing the tool to move (retract) along the axis for which a retract amount is specified for parameter No. 7741. After the end of retraction, the retract completion signal RTRCTF is output. This function is intended to retract the tool from the workpiece immediately when a tool breakage is detected.

- The feedrate assumed during retract is the same as that set in parameter No. 7740. A feedrate override is invalid.
- A feed hold is invalid to movement during retraction.
- If the retract signal is set to 1 during automatic operation, a retract operation is performed and automatic operation is stopped.
- The retract completion signal becomes 0 when the tool has moved along one of the retract axes.



Signal

Retract signal RTRCT<Gn066.4>

[Classification] Input signal

[Function] Causes the tool to rectract along the axis for which a retract amount is set in the parameter.

[Operation] When this signal becomes 1, the CNC operates as described below.

- Captures the rise of the signal and causes the tool to retract along the axis for which a retract amount is set in parameter No. 7741. The retract amount and the retract speed assume the values previously set in parameters Nos. 7741 and 7740.
  After the end of retraction, the retract completion signal RTRCTF becomes 1.
  The retract signal is effective in either automatic operation mode (MEM, MDI) or manual mode (HNDL, JOG, etc.).
  If the retract signal is set to 1 during automatic operation, a retract operation is performed and automatic operation is stopped.

Retract completion signal RTRCTF<Fn065.4>

[Classification] Output signal

[Function] Notifies that retraction is completed.

[Output cond.] This signal becomes 1 when:

- A retract operation is completed.
  If a retract operation is performed on two or more axes, the operation is completed on all retract axes.

It becomes 0 when:

- After the end of a retract operation, a move command is issued for one of the retract axes.

- Follow-up occurs due to a servo-off, emergency stop, or other conditions.

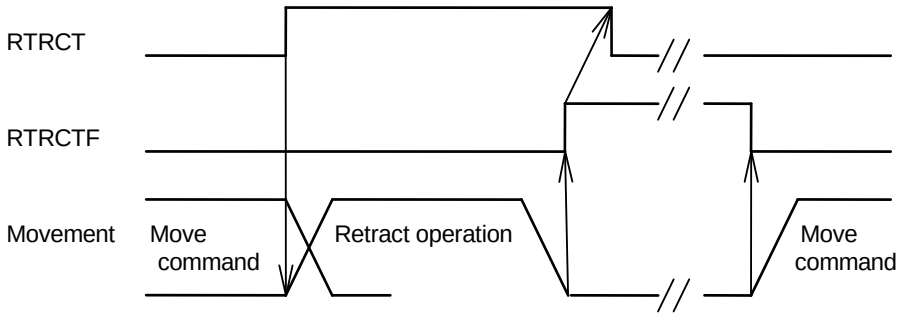
NOTE
While the retract completion signal is 1, the retract signal cannot be accepted.

Signal address

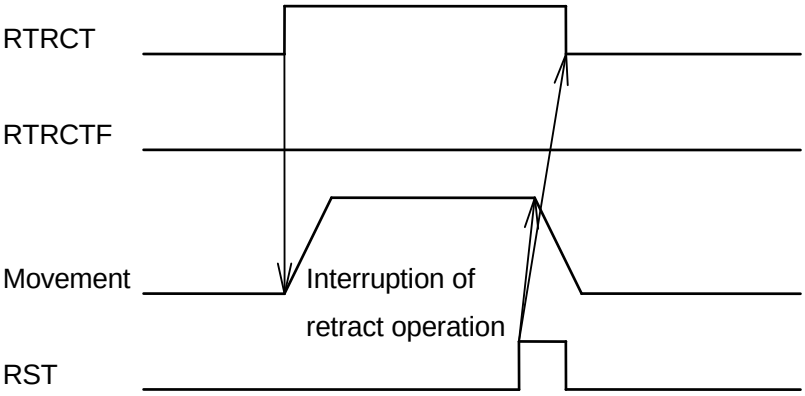
	#7	#6	#5	#4	#3	#2	#1	#0
Gn066				RTRCT				
Fn065				RTRCTF				

Timing chart

(1) RTRCT and RTRCTF on/off timing

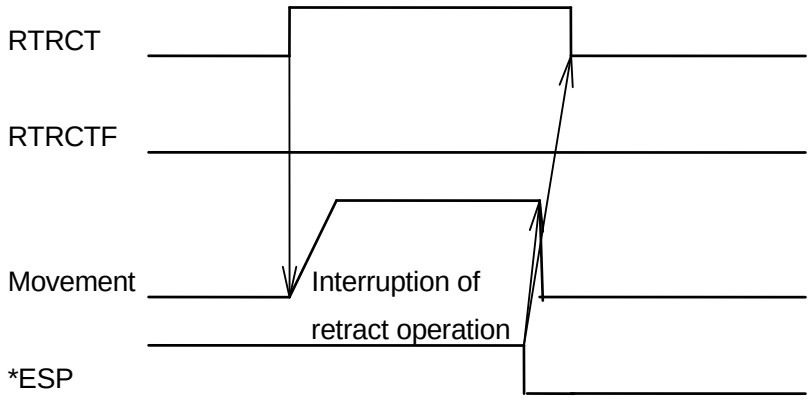


(2) Interruption of retraction due to a reset



Turn the RTRCT signal off at the same time as turning the RST signal on.

(3) Interruption of retraction due to an emergency stop



Turn RTRCT off at the same time as turning *ESP off.

Parameter

7740	Retract speed

[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 feedrate to be assumed during retraction for each axis.

7741	Retract amount

[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 retract amount for each axis.

NOTE
Movement (retraction) is effected by the set value, regardless of which of a diameter and radius is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
7704								ACR

[Input type] Parameter input
[Data type] Bit path

#0 ACR In the AI contour control/AI advanced preview control (M series), or advanced preview control mode (T series), general purpose retract operation is:
0: Not used.
1: Used.

7745	Time constant for linear acceleration/deceleration during retraction for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 1000

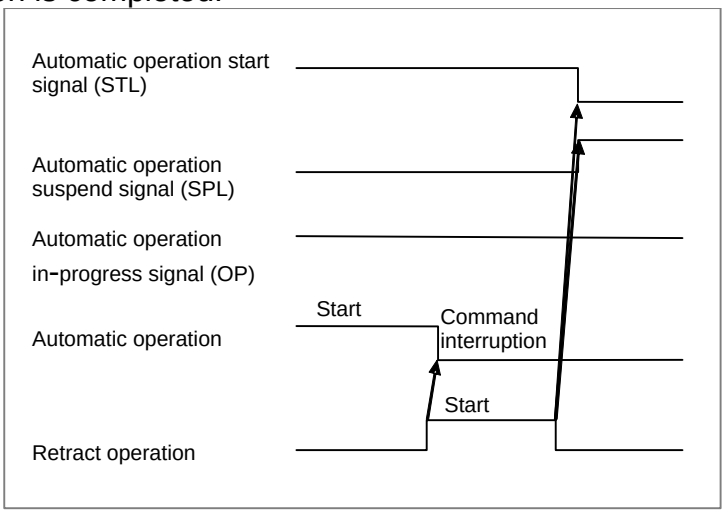
This parameter is used to set the acceleration for linear acceleration/deceleration during retraction with the general purpose retract function. For each axis, set the time required until the speed set in parameter No. 7740 is attained (time constant).

NOTE
This parameter is effective when ACR, bit 0 of parameter No. 7704, is 1.

Caution

⚠ CAUTION

1 A feedrate override is invalid to the retract speed.
2 During a retract operation, an interlock to a retract axis is effective.
3 During a retract operation, a machine lock to a retract axis is effective. The retract operation is completed in the machine lock state, and the retract completion signal is output.
4 During retraction, a feed hold is invalid.
5 The retract direction is the movement direction of the machine regardless of whether a mirrored image (signal and setting) is valid or not. (A mirror image is not applied to the updating of absolute coordinates.)
6 When retraction is performed during automatic operation, the execution of a command is interrupted as soon as a retract operation starts. However, the operation state is switched to the automatic operation stopped state when the retract operation is completed.



7 Retract movement is effected with non-linear type positioning.

⚠ CAUTION

8 During a retract operation, acceleration/deceleration assumes the acceleration/deceleration state at the start of retraction. In AI contour control mode, acceleration/deceleration before interpolation is not applied to a retract operation. By setting 1 in ACR, bit 0 of parameter No. 7704, and setting a retract time constant in parameter No. 7745, it is possible to apply linear acceleration/deceleration to a retract operation in AI contour control mode.

9 During a retract operation, performing a reset or emergency stop causes the operation to be interrupted. In this case, the retract completion signal does not become 1.

10 If, during a retract operation, a servo alarm, a retract axis OT alarm, or a PW alarm is generated, the retract operation is interrupted. In this case, the retract completion signal does not become 1. With alarms other than a PW alarm, an OT alarm, and a servo alarm, the retract operation is not interrupted.

11 After the start of a retract operation, setting the retract signal RTRCT to 0 does not cause the retract operation to be interrupted.

12 While the retract completion signal RTRCTF is 1, the retract signal RTRCT cannot be accepted.

13 Even while a threading command is being executed, the retract function is effective. When the retract function is executed, the threading operation is stopped immediately, and a retract operation is started.

14 Even in a canned cycle (during tapping and during rigid tapping), the retract function is effective. When the retract function is executed, the cycle operation is stopped immediately, and a retract operation is started.

15 When retraction is performed on an axis on which movement is effected with PMC axis control, the move command for the PMC axis is not canceled. If, therefore, retraction is to be performed on a PMC axis on which movement is effected, set the retract signal RTRCT to 1 and use the PMC axis reset signal ECLRg to cancel PMC axis control.

16 If an electronic gear box option (M series) is attached, this function is invalid.

17 This function cannot be used for axes related to the functions given below.
- Synchronous/Composite control (T series)
- Superimposed Control (T series)
- Axis synchronous control
- Arbitrary angular axis control

7 FEEDRATE CONTROL/ACCELERATION AND DECELERATION CONTROL

Chapter 7, "FEEDRATE CONTROL/ACCELERATION AND DECELERATION CONTROL", consists of the following sections and subsections:

7.1 FEEDRATE CONTROL 496
7.1.1 Rapid Traverse Rate 497
7.1.2 Cutting Feedrate Clamp 499
7.1.3 Feed Per Minute 499
7.1.4 Feed Per Revolution/Manual Feed Per Revolution 501
7.1.5 One-digit F Code Feed (M Series) 502
7.1.6 Inverse Time Feed (M Series) 505
7.1.7 Override 505
7.1.8 Automatic Corner Override (M Series) 511
7.1.9 External Deceleration 516
7.1.10 Acceleration-Based Speed Control in Circular Interpolation 521
7.1.11 High Precision and High Speed Functions (Advanced Preview Control (T Series)/ AI Advanced Preview Control (M Series)/AI Contour Control (M Series)/AI Contour Control II (M Series)) 522
7.1.12 Speed Command Extension in Least Input Increment C 552
7.2 ACCELERATION/DECELERATION CONTROL 555
7.2.1 Automatic Acceleration/Deceleration 555
7.2.2 Rapid Traverse Bell-shaped Acceleration/Deceleration 560
7.2.3 Linear Acceleration/Deceleration after Cutting Feed Interpolation 562
7.2.4 Bell-Shaped Acceleration/Deceleration after Cutting Feed Interpolation (M Series) 565
7.2.5 Corner Control 567
7.2.6 Feed Forward in Rapid Traverse 572
7.3 JERK CONTROL (M Series) 572
7.3.1 Speed Control with Change of Acceleration on Each Axis 572
7.3.2 Look-Ahead Smooth Bell-Shaped Acceleration/Deceleration before Interpolation 576

7.1 FEEDRATE CONTROL

The feed functions control the feedrate of the tool. The following two feed functions are available:

1. Rapid traverse
   When the positioning command (G00) is specified, the tool moves at a rapid traverse rate set in the CNC (parameter No. 1420).
2. Cutting feed
   The tool moves at a programmed cutting feedrate.

Override can be applied to a rapid traverse rate or cutting feedrate using the override signal.

7.1.1 Rapid Traverse Rate

Overview

The positioning command (G00) positions the tool by rapid traverse.

G00 IP_ ;

G00 : G code (group 01) for positioning (rapid traverse)
IP_ ; Dimension word for the end point

In rapid traverse, the next block is executed after the specified rate becomes 0 and the servo motor reaches a certain range set by the parameter (No.1826) (in-position check).
A rapid traverse rate is set for each axis by parameter No. 1420, so no rapid traverse rate need be programmed.
The following overrides can be applied to a rapid traverse rate with the rapid traverse override signal:F0, 25, 50, 100%
F0: Allows a fixed feedrate to be set for each axis by parameter No. 1421.
The 1% step rapid traverse override selection signal or 0.1% step rapid traverse override selection signal enables a rapid traverse override to be applied in steps of 1% or 0.1% over a range of 0% to 100%.

Signal

Rapid traversing signal RPDO<Fn002.1>

[Function] This signal indicates that a move command is executed at rapid traverse.
[Output cond.] "1" indicates that an axis starts moving after rapid traverse has been selected in automatic operation mode or manual operation mode.
"0" indicates that an axis starts moving after a feedrate other than rapid traverse has been selected in automatic operation mode or manual operation mode.

NOTE
1 The rapid traverse in automatic operation includes all rapid traverses in canned cycle positioning, automatic reference point return, etc., as well as the move command G00. The manual rapid traverse also includes the rapid traverse in reference position return.
2 Once rapid traverse has been selected and movement is started, this signal remains "1", including during a stop, until another feedrate has been selected and movement is started.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn002							RPDO	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401		RDR		RF0			LRP	

[Input type] Parameter input
[Data type] Bit path

#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.

#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.

#6 RDR Dry run for rapid traverse command
- 0: Disabled
- 1: Enabled

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.

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 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.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Rapid traverse

7.1.2 Cutting Feedrate Clamp

Overview

An upper limit can be set on the cutting feedrate along each axis (parameter No. 1430). If an actual cutting feedrate (with an override applied) exceeds a specified upper limit, it is clamped to the upper limit.

Parameter

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.
In cutting feed, the maximum feedrate is clamped so that the feedrate of each axis does not exceed its maximum cutting feedrate.

Alarm and message

Number	Message	Description
PS5009	PARAMETER ZERO (DRY RUN)	The dry run feedrate parameter No. 1410 or maximum cutting feedrate parameter No. 1430 for each axis has been set to 0.
PS5011	PARAMETER ZERO (CUT MAX)	The maximum cutting feedrate parameter No. 1430 has been set to 0.

Warning

⚠ WARNING
CNC calculation may involve a feedrate error of ±2% with respect to a specified value. However, this is not true for acceleration/deceleration. To be more specific, this error is calculated with respect to a measurement on the time the tool takes to move 500 mm or more in the steady state.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Cutting feed

7.1.3 Feed Per Minute

Overview

- Feed per minute

After specifying G94 (G98 for T series) (in the feed per minute mode), the amount of feed of the tool per minute is specified by setting a number after F. G94 (G98 for T series) is a modal code. Once a G94 (G98 for T series) is specified, it is valid until G95 (G99 for lathe system) (feed per revolution) is specified. At power-on, the feed per minute mode (feed per revolution mode for lathe system) is set.
An override from 0% to 254% (in 1% steps) can be applied to feed per minute with the feedrate override signal.

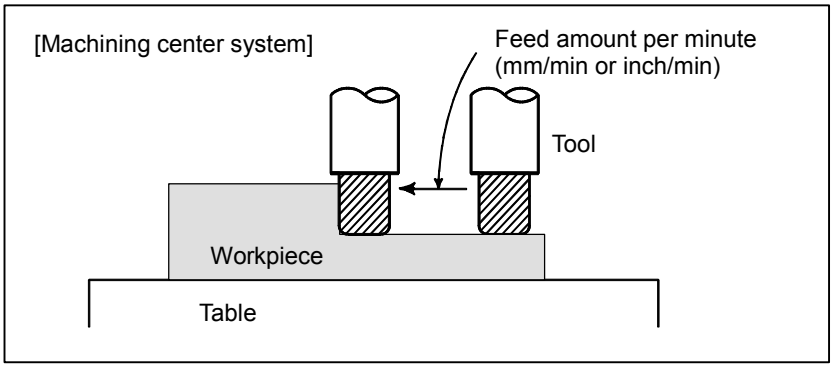


Fig. 7.1.3 (a) Feed per minute

CAUTION
No override can be used for any commands such as for threading.

Format

M

Feed per minute

G94; G code for feed per minute (Group 05)
F_; Feed rate (mm/min or inch/min)

T

Feed per minute

G98; G code for feed per minute (Group 05)
F_; Feed rate (mm/min or inch/min)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3402				FPM				

[Input type] Parameter input
[Data type] Bit path

#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.
For information about the clear state, refer to the appendix of the Operatorr's Manual (B-64304EN).

Alarm and message

Number	Message	Description
PS0011	FEED ZERO (COMMAND)	The cutting feedrate instructed by an F code has been set to 0. 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.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Cutting feed

7.1.4 Feed Per Revolution/Manual Feed Per Revolution

Overview

- Feed per revolution

After specifying G95 (G99 for T series) (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. G95 (G99 for T series) is a modal code. Once a G95 is specified, it is valid until G94 (G98 for T series) (feed per minute) is specified.

An override of between 0 and 254% (in steps of 1%) can be applied to feed per revolution, using the feedrate override signals (*FV0 to *FV7).

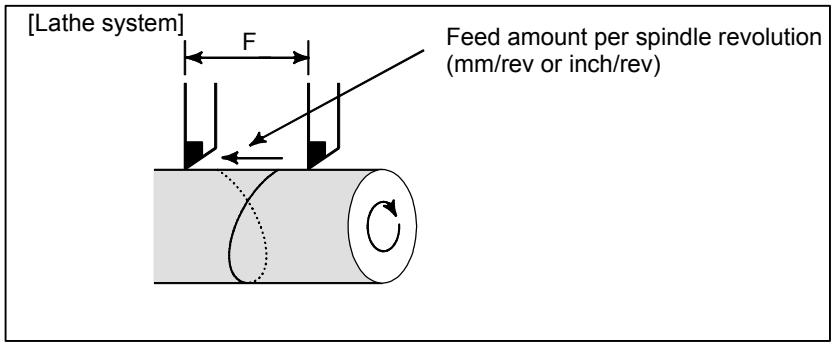


Fig. 7.1.4 (a) Feed per revolution

- Manual feed per revolution

Jog feedrate can be specified by feed per revolution.

Parameter

	#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

NOTE
1 When using the position coder, set this parameter to 0.
2 While this parameter is set to 1, threading cannot be performed even if a position coder is provided.

#4 JRV Jog feed or incremental feed is
0: Performed at feed per minute.
1: Performed at feed per revolution.

NOTE
Specify a feedrate in parameter No.1423.

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 JRV, bit 4 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 JRV, bit 4 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).

CAUTION

⚠ CAUTION
When the speed of the spindle is low, feedrate fluctuation may occur.
The slower the spindle rotates, the more frequently feedrate fluctuation occurs.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Cutting feed

7.1.5 One-digit F Code Feed (M Series)

M

Overview

- One-digit F code feed

When a one-digit number from 1 to 9 is specified after F, the feedrate set for that number in a parameter (Nos. 1451 to 1459) is used. When F0 is specified, the rapid traverse rate is applied.

NOTE
To use one-digit F code feed, set 1 in bit 1 (F1D) of parameter No. 8131.

The feedrate corresponding to the number currently selected can be increased or decreased by turning on the switch for changing F1-digit 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:

$$\Delta F = \frac{Fmaxi}{100n} (i = 1, 2)$$

Fmax : Feedrate upper limit for F1-F4 set by parameter 1460, or
feedrate upper limit for F5-F9 set by parameter 1461
n : Any value of 1 to 127 set by parameter 1450

The feedrate set or altered is kept even while the power is off. The current feedrate is displayed on the screen.

- Cutting feed override during one-digit F code feed

The following functions are enabled for a cutting feedrate during one-digit F code feed:

- Feedrate override
- Override cancel

Even during machining using one-digit F code feed, the feedrate can be freely changed, within an applicable override range, from the parameter-set value by changing the override switch.

Signal

One-digit F code feed signal F1D<Gn016.7>

[Classification] Input signal

[Function] Increases or decreases one-digit F speed set by the parameters No. 1451 to 1459 using the manual pulse generator.
Since the manual pulse generator may also be used for axis feeding, signal F1D <G016.7> designates which function may be used.

[Operation] When the signal is "1", the one-digit F speed can be increased/decreased using the manual pulse generator.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G016	F1D							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1406								
	F1O							

[Input type] Parameter input
[Data type] Bit path

#7 F1O For the cutting feedrate specified by a single-digit F code (F1 to F9), feedrate override and override cancellation are:
0: Disabled.
1: Enabled.

> **NOTE**
> For the F0 feedrate, rapid traverse override is enabled regardless of the setting of this parameter.

1450	
	Number of revolution of manual pulse generator to reach maximum feedrate

[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 code feed.
$\Delta F=Fmaxi/100n$ (where, i=1 or 2)

In the above equation, n is, the number of revolutions of the manual pulse generator, required to reach feedrate Fmaxi. Fmaxi refers to the upper limit of the feedrate for an 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 1460)
Fmax2: Upper limit of the feedrate for F5 to F9 (parameter 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)
Set Feedrates for one-digit F code feed commands F1 to F9.
When an one-digit F code feed command is executed, as the feedrate is changed by turning the manual pulse generator, these parameter values also change accordingly.

1460	Upper feedrate limit for F1 to F4
1461	Upper feedrate limit 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 an one-digit F code feed command F1 to F4 is executed, the upper limit is that set in parameter 1460. If an one-digit F code feed command F5 to F9 is executed, the upper limit is that set in parameter 1461.

	#7	#6	#5	#4	#3	#2	#1	#0
8131								
							F1D	

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#1 F1D One-digit F code feed is:
0: Not Used.
1: Used.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Cutting feed

7.1.6 Inverse Time Feed (M Series)

M

Overview

Feedrate of the tool can be specified by the move distance of the block and inverse time (FRN).

- Linear interpolation (G01)

FRN=1/Time (min) = Speed/Distance

Speed: mm/ min (metric input)
inch/ min (inch input)

Distance: mm (metric input)
inch (inch input)

- Circular interpolation (G02, G03)

FRN=1/Time (min) = Speed/Circle radius

Speed: mm/ min (metric input)
inch/ min (inch input)

Circle radius: mm (metric input)
inch (inch input)

> ⚠ **CAUTION**
> In circular interpolation, the distance is not an actual distance of the block but the speed is calculated from the circle radius.

Alarm and message

Number	Message	Description
PS0011	FEED ZERO (COMMAND)	The cutting feedrate instructed by an F code has been set to 0. 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.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Cutting feed

7.1.7 Override

7.1.7.1 Rapid traverse override

Overview

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).

The 1% step rapid traverse override selection signal or 0.1% step rapid traverse override selection signal enables a rapid traverse override to be selected in steps of 1% or 0.1% over a range of 0% to 100%.

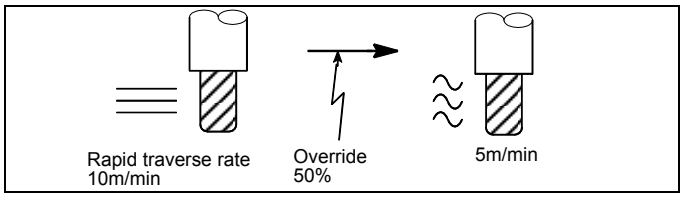


Fig. 7.1.7 (a) Rapid traverse override

- Feedrate

Actual feedrate is obtained by multiplying the rapid traverse rate preset by parameter No.1420 by the override value determined by this signal, whether in automatic or manual operation (including manual reference position return).

- F0 rate

For F0 value, an absolute value is set by parameter No.1421 within a range of 0 to rapid traverse rate (for each axis).

- 1% step rapid traverse override

1% step rapid traverse override selection signal HROV determines whether rapid traverse override specified with rapid traverse override signals ROV1 and ROV2 is used or 1% step rapid traverse override is used.
When signal HROV is 0, override is applied to the rapid traverse rate using signals ROV1 and ROV2.
When signal HROV is 1, ROV1 and ROV2 are ignored, 1% step rapid traverse override signals *HROV0 to *HROV6 being used to override the rapid traverse rate.

- 0.1% step rapid traverse override

With the 0.1% step rapid traverse override selection signal FHROV, a 1% step rapid traverse override or 0.1% step rapid traverse override can be selected for use.
When the HROV signal is set to 1 and the FHROV signal is set to 0, a 1% step rapid traverse override is applied.
When the HROV and FHROV signals are both set to 1, the 1% step rapid traverse override is disabled, and the rapid traverse override specified by the 0.1% step rapid traverse override signals *FHRO0 to *FHRO9 is enabled.

- PMC axis control

These 1% step rapid traverse override signals or 0.1% step rapid traverse override selection signals are also effective to the rapid traverse rate for the PMC axis. When rapid traverse override is applied to the PMC axis (using signals ROV1E and ROV2E) with the setting of the OVE bit (bit 2 of parameter No. 8001) independently of the CNC, the 1% step rapid traverse override signals are ineffective.

Signal

Rapid traverse override signal ROV1,ROV2<Gn014.0 to 1>

[Classification] Input signal
[Function] These signals override the rapid traverse rate
[Operation] These code signals correspond to the rates as follows (Table 7.1.7 (a)):

Table 7.1.7 (a) Signals ROV1 to ROV2 and override value

Rapid traverse override		Override value
ROV2	ROV1	
0	0	100%
0	1	50%
1	0	25%
1	1	F0

F0 : Set in parameter No. 1421

1% step rapid traverse override selection signal HROV<Gn096.7>

[Classification] Input signal

[Function] Selects the rapid traverse override signals or the 1% step rapid traverse override signals.

[Operation] When this signal is set to 1, the values of *HROV0 to *HROV6 are enabled. (Rapid traverse override by signals ROV1 and ROV2 is disabled.)
When this signal is set to 0, Rapid traverse override by signals ROV1 and ROV2 is enabled. (the values of *HROV0 to *HROV6 are disabled.)

1% step rapid traverse override signals *HROV0 to *HROV6<Gn096.0 to 6>

[Classification] Input signal

[Function] Applies override to the rapid traverse rate in the range of 0% to 100% in steps of 1%.

[Operation] These seven signals give a binary code indicating an override applied to the rapid traverse rate.

- When a binary code corresponding to an override value of 101% to 127% is specified, the applied override is clamped at 100%.
- Signals *HROV0 to *HROV6 are inverted signals.
  To set an override value of 1%, set signals *HROV0 to *HROV6 to 1111110, which corresponds to a binary code of 0000001.

0.1% step rapid traverse override selection signal FHROV<Gn353.7>

[Classification] Input signal

[Function] Selects whether 1% rapid traverse override signals or 0.1% step rapid traverse override signals are enabled when the HROV signal is set to 1.

[Operation] When this signal is set to 1, the values of *FHRO0 to *FHRO9 are enabled. (The values of the *HROV0 to *HROV6 signals are invalidated.)
When this signal is set to 0, the values of *HROV0 to *HROV6 are enabled. (The values of the *FHRO0 to *FHRO9 signals are invalidated.)

0.1% step rapid traverse override signals *FHRO0 to *FHRO9<Gn352.0 to 7, Gn353.0 to 1>

[Classification] Input signal

[Function] Applies override to the rapid traverse rate in the range of 0% to 100% in steps of 0.1%.

[Operation] These ten signals give a binary code indicating an override applied to the rapid traverse rate.

- When a binary code corresponding to an override value of 100.1 to 102.3% is specified, the applied override is clamped at 100%.
- Signals *FHRO0 to *FHRO9 are inverted signals.
  To set an override value of 0.1%, set signals *FHRO0 to *FHRO9 to 1111111110, which corresponds to a binary code of 0000000001.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn014							ROV2	ROV1
Gn096	HROV	*HROV6	*HROV5	*HROV4	*HROV3	*HROV2	*HROV1	*HROV0
Gn352	*FHRO7	*FHRO6	*FHRO5	*FHRO4	*FHRO3	*FHRO2	*FHRO1	*FHRO0
Gn353	FHROV						*FHRO9	*FHRO8

Parameter

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.

	#7	#6	#5	#4	#3	#2	#1	#0
3002				IOV				

[Input type] Parameter input
[Data type] Bit path

#4 IOV Override-related signal logic is:
0: Used without modification
(A signal of negative logic is used as a negative logic signal, and a signal of positive logic is used as a positive logic signal.)
1: Inverted
(A signal of negative logic is used as a positive logic signal, and a signal of positive logic is used as a negative logic signal.)

The signals indicated below are affected.
Signal of negative logic:
Feedrate override signals *FV0 to *FV7<Gn012>
Feedrate override signals (for PMC axis control)
*EFOV0g to *EFOV7g<G0151/G0163/G0175/G0187>
Software operator's panel signals *FV0O to *FV7O<Fn078>
Signals of positive logic:
Rapid traverse override signals ROV1,ROV2<Gn014.0 to 1>
Software operator's panel signals ROV1O,ROV2O<Fn076.4 to 5>
Rapid traverse override signals (for PMC axis control)
EROV1g,EROV2g<G0150.0 to 1, G0162.0 to 1, G0174.0 to 1, G0186.0 to 1>
The signals below are not affected.
1% step rapid traverse override selection signal
HROV<Gn096.7>
1% step rapid traverse override signals
*HROV0 to *HROV6<Gn096.0 to 6>
0.1% step rapid traverse override selection signal
FHROV<Gn353.7>
0.1% step rapid traverse override signals
*FHRO0 to *FHRO9<Gn352.0 to 7, Gn353.0 to 1>

	#7	#6	#5	#4	#3	#2	#1	#0
8001						OVE		

[Input type] Parameter input
[Data type] Bit path

#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	No.8001#2=0 (same signals as those used for the CNC)		No.8001#2=1 (signals specific to the PMC)	
Feedrate override signals	*FV0 to *FV7	<Gn012>	*EFV0g to *EFV7g	<G0151>
Override cancellation signal	OVC	<Gn006.4>	EOVCg	<G0150.5>
Rapid traverse override signals	ROV1, ROV2	<Gn014.0 to 1>	EROV1g,EROV2g	<G0150.0 to 1>
Dry run signal	DRN	<Gn046.7>	EDRNg	<G0150.7>
Rapid traverse selection signal	RT	<Gn019.7>	ERTg	<G0150.6>
1% step rapid traverse override selection signal	HROV	<Gn096.7>	None	
1% step rapid traverse override signals	*HROV0 to *HROV6	<Gn096.0 to 6>	None	
0.1% step rapid traverse override selection signal	FHROV	<Gn353.7>	None	
0.1% step rapid traverse override signals	*FHRO0 to *FHRO9	<Gn352.0 to 7, Gn353.0 to 1>	None	

(The signal addresses at PMC selection time are for the first group.)

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Rapid traverse override

7.1.7.2 Feedrate override

Overview

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.

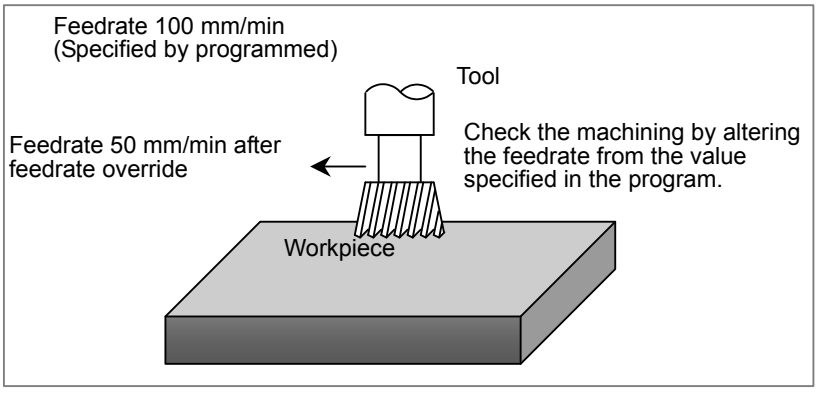


Fig. 7.1.7 (b) Feedrate override

Signal

Feedrate Override signal *FV0 to *FV7<Gn012>

[Classification] Input signal

[Function] These signals override the cutting feedrate. Eight binary code signals correspond to override values as follows:

Override value= $\sum_{i=0}^{7} | 2^i \times Vi | \%$

Vi=0 when *FVi is "1" and
Vi=1 when *FVi is "0"
These signals have the following weight.
*FV0 : 1% *FV1 : 2%
*FV2 : 4% *FV3 : 8%
*FV4 : 16% *FV5 : 32%
*FV6 : 64% *FV7 : 128 %
When all signals are "0", they are regarded as overriding 0% in the same way as when all signals are "1" .
Thus, the override is selectable in steps over a range of 0 to 254%.

[Operation] Actual feedrate is obtained by multiplying the specified speed by the override value selected by this signal.
The override is regarded as 100%, regardless of this signal, in the following cases:
- Override cancel signal OVC<Gn006.4> is "1".
- During cutting in tap cycle of canned cycle;
- Tapping mode (G63)
- Thread cutting is in progress.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn012	*FV7	*FV6	*FV5	*FV4	*FV3	*FV2	*FV1	*FV0

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401				RF0				

[Input type] Parameter input
[Data type] Bit path

#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
3002				IOV				

[Input type] Parameter input
[Data type] Bit path

#4 IOV Override-related signal logic is:
0: Used without modification
(A signal of negative logic is used as a negative logic signal, and a signal of positive logic is used as a positive logic signal.)
1: Inverted
(A signal of negative logic is used as a positive logic signal, and a signal of positive logic is used as a negative logic signal.)

The signals indicated below are affected.
Signal of negative logic:
Feedrate override signals *FV0 to *FV7<Gn012>
Feedrate override signals (for PMC axis control)
*EFOV0g to *EFOV7g<G0151/G0163/G0175/G0187>

Software operator's panel signals *FV0O to *FV7O<Fn078>
Signals of positive logic:
Rapid traverse override signals ROV1,ROV2<Gn014.0 to 1>
Software operator's panel signals ROV1O,ROV2O<Fn076.4 to 5>
Rapid traverse override signals (for PMC axis control)
EROV1g,EROV2g<G0150.0 to 1, G0162.0 to 1, G0174.0 to 1, G0186.0 to 1>
The signals below are not affected.
1% step rapid traverse override selection signal
HROV<Gn096.7>
1% step rapid traverse override signals
*HROV0 to *HROV6<Gn096.0 to 6>
0.1% step rapid traverse override selection signal
FHROV<Gn353.7>
0.1% step rapid traverse override signals
*FHRO0 to *FHRO9<Gn352.0 to 7, Gn353.0 to 1>

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Cutting feed

7.1.7.3 Override cancel

Overview

The override cancel signal fixes the feedrate override to 100%.

Signal

Override cancel signal OVC<Gn006.4>

[Classification] Input signal
[Function] Feedrate override is fixed to 100%.
[Operation] When the signal is "1", the CNC operates as follows:
- The feedrate override is fixed to 100% regardless of the feedrate override signals or second feedrate override signals. That is, the specified feedrate is used as is.
- Rapid traverse override and spindle speed override are not affected.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn006				OVC				

7.1.8 Automatic Corner Override (M Series)

M

The movement of a tool in an inner corner or an inner arc is automatically decelerated during cutter compensation, so that the load on the tool is reduced, and a surface obtained by cutting becomes smooth.

NOTE
To use automatic corner override, set 1 in bit 3 (AOV) of parameter No. 8131.

7.1.8.1 Inner corner automatic override (G62)

Explanation

- Overriding conditions

When G62 is specified, and the tool path with cutter 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. 7.1.8 (a)).

$2° \leq \theta \leq \theta p \leq 178°$ in Fig. 7.1.8

θp 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.

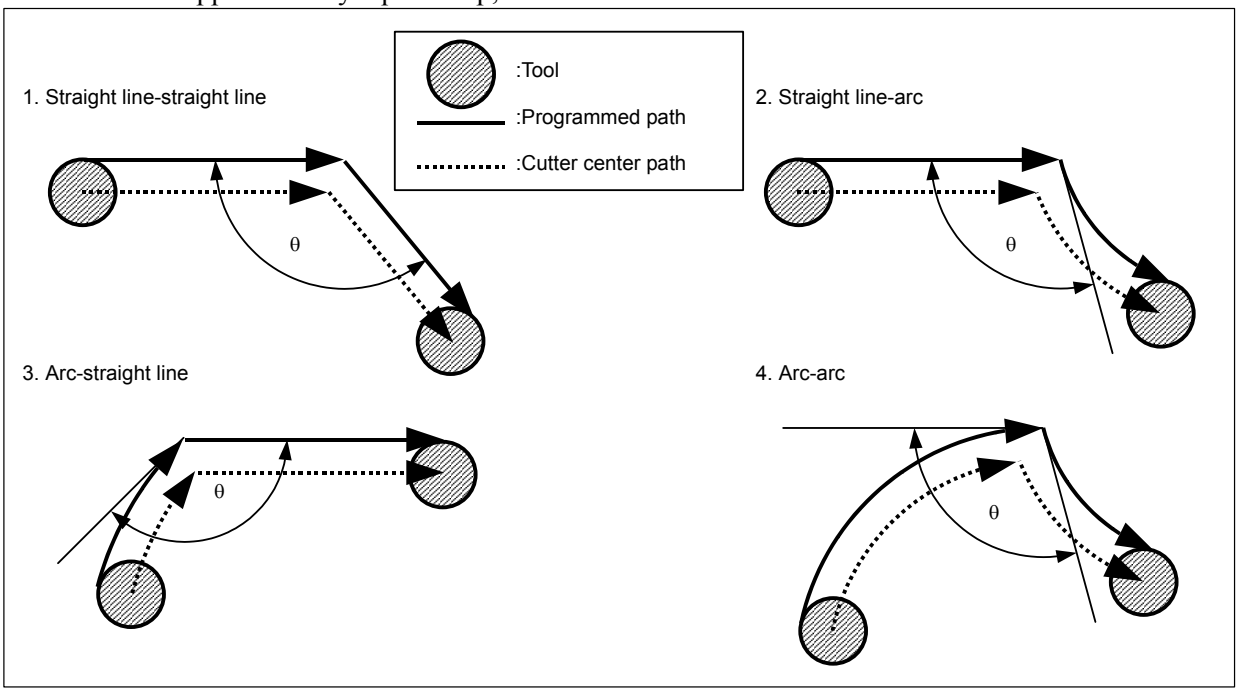


Fig. 7.1.8 (a) Inner corner

- Overriding range

When an inner corner is found, the feedrate is overridden before and after the intersection of the corresponding tool paths. Feedrate override distances Le and Ls are linear dimensions between the intersection of the tool paths and a point on either cutter center path (Fig. 7.1.8 (b), Fig. 7.1.8 (c), Fig. 7.1.8 (d)). Le and Ls are specified in parameters No.1713 and No.1714.

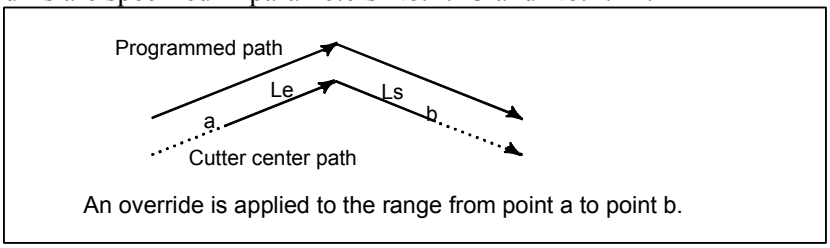


Fig. 7.1.8 (b) Overriding range (straight line - straight line)

An arc-shaped path is overridden if the start point and end point of the arc is in the same quadrant or in two adjacent quadrants (Fig. 7.1.8 (c)).

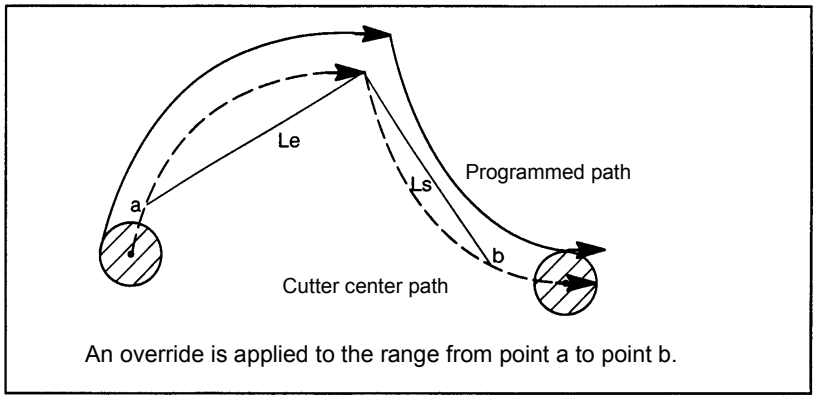


Fig. 7.1.8 (c) Overriding range (arc - arc)

On arc-shaped path <2> programmed as indicated below, an override occurs between points a and b and between points c and d (Fig. 7.1.8 (d)).

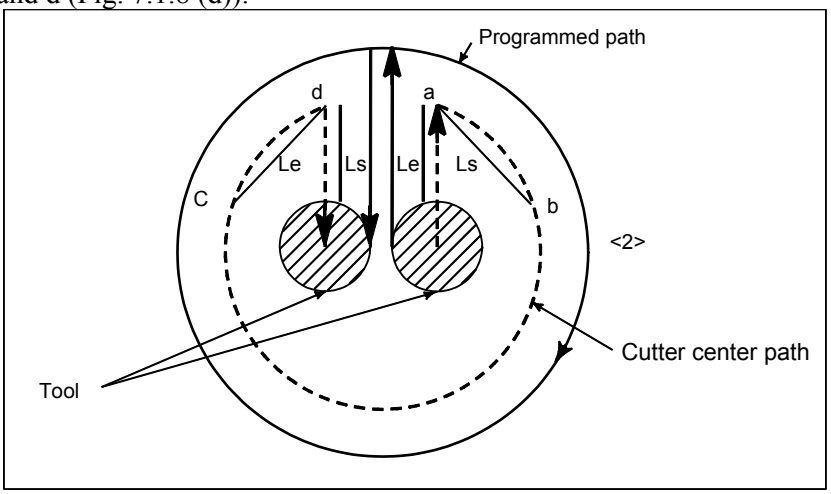


Fig. 7.1.8 (d) Overriding range (straight line - arc, arc - straight line)

- Override value

An override value is set with parameter No. 1712. An override value is valid even for dry run and F1-digit feed specification.
In the feed per minute mode, the actual feedrate is as follows:
F × (inner corner automatic override) × (feedrate override)

Limitation

- Acceleration/deceleration before interpolation

The inner corner override function is disabled during acceleration/deceleration before interpolation.

- Start-up, G41, G42

The inner corner override function is not carried out for a corner preceded by a cutter compensation start-up block or a corner followed by a block containing G41 or G42,.

- Offset value

If the offset value is zero, the inner corner override function is not executed.

7.1.8.2 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. 7.1.8 (e)). This function is valid in the cutter compensation mode, regardless of the G62 code.

$F \times \frac{Rc}{Rp}$

Rc: Cutter center path radius
Rp: Programmed tool path radius

It is also valid for the dry run and the one digit F cord feed.

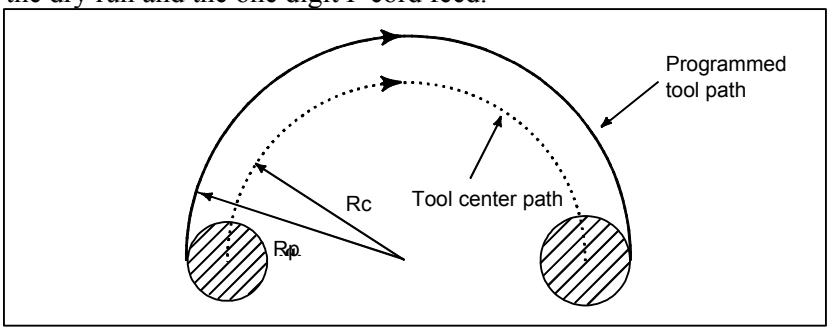


Fig. 7.1.8 (e) Internal circular cutting feedrate change

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). When 0 is set in parameter No. 1710, the minimum deceleration ratio (MDR) is 100%.

> ⚠ **CAUTION**
> When internal circular cutting must be performed together with automatic override for inner corners, the feedrate of the tool is as follows:
> F×Rc/Rp×(inner corner override)×(feedrate override)

Parameter

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

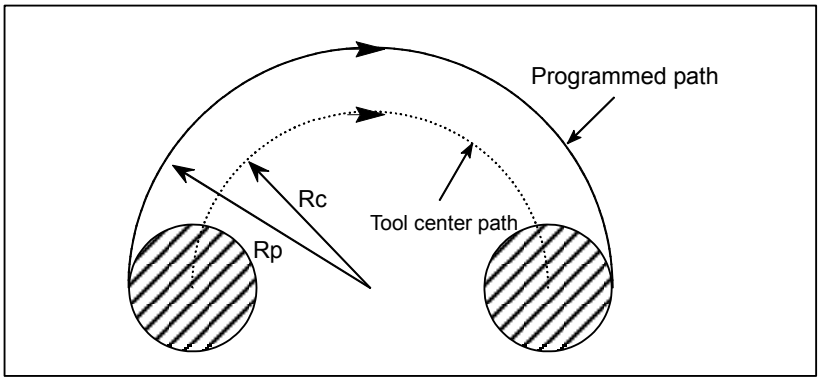
Set a minimum deceleration ratio (MDR) for an inner circular cutting feedrate change by automatic corner override.

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≒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.

NOTE
When this parameter is set to 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] 0 to 100
Set an inner corner override value in automatic corner overriding.

1713	
	Start distance (Le) for inner corner override

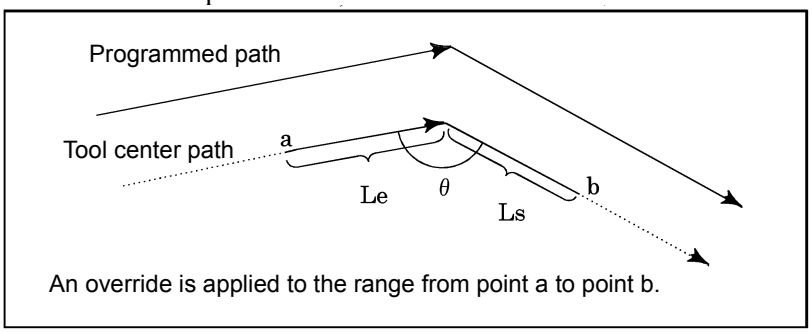
[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Min. unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 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 parameter No. 1713 and No. 1714.



	#7	#6	#5	#4	#3	#2	#1	#0
8131								
					AOV			

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#3 AOV Automatic corner override is:
0: Not Used.
1: Used.

7.1.9 External Deceleration

Overview

The control axis is externally decelerated. The feedrate is decelerated by the external deceleration signals from the machine. The deceleration rate is set by the parameters.
The external deceleration signal are provided for each axis and direction.
As three types of deceleration condition settings can be dynamically selected by the signals.
External deceleration can be applied to rapid traverse rate, cutting feedrate, and manual handle feedrate.

NOTE
To use external deceleration, set 1 in bit 2 (EDC) of parameter No. 8131.

- Rapid traverse and cutting feed

Three types of deceleration conditions can be set by the parameters.

When deceleration conditions specified by multiple external signals input during machining are conflicted, the condition with the lowest external deceleration rate is applied.
External deceleration settings 2 and 3 can be switched between enabled and disabled by the parameters.

- Positioning of linear interpolation type

When positioning of linear interpolation type is used (When 1 is set in bit 1 (LRP) of parameter No.1401), the external deceleration rate selected by bit 5 (EDR) of parameter No.1405 is used.
If the external deceleration rate for cutting feed is selected (If 0 is set in parameter EDR), be sure to set 1 to bit 4 (EDPx) of parameter No.1005 and bit 5 (EDMx) of parameter No.1005.
When 0 is set in these parameters, the external deceleration becomes invalid.

- Manual handle feed

Three types of deceleration conditions can be set by the parameters. For handle feed, the maximum feedrate is switched when the external deceleration signal in the positive or negative direction for the handle axis is set to 0. When multiple conditions are conflicted, the condition with the lowest maximum feedrate is applied.

- PMC axis control

For information about the external deceleration function using PMC axis control, see Chapter 16, "PMC CONTROL FUNCTION".

Signal

External deceleration signals 1
*+ED1 to *+ED5<Gn118.0-Gn118.4>,*-ED1 to *-ED5<Gn120.0-Gn120.4>
External deceleration signals 2
*+ED21 to *+ED25<Gn101.0-Gn101.4>,*-ED21 to *-ED25<Gn103.0-Gn103.4>
External deceleration signals 3
*+ED31 to *+ED35<Gn107.0-Gn107.4>,*-ED31 to *-ED35<Gn109.0-Gn109.4>

[Classification] Input signal

[Function] These signals specifies whether to apply external deceleration to each direction for each control axis. The sign + or - preceding ED indicates the direction of feed which is decelerated. The number following ED indicates the number of external deceleration setting. The last number indicates the number of a control axis.

* xEDyz

z : 1 ….. The 1st axis is decelerated.
2 ….. The 2nd axis is decelerated.
3 ….. The 3rd axis is decelerated.
:

y : None.. External deceleration setting 1 is selected.
2 ….. External deceleration setting 2 is selected.
3 ….. External deceleration setting 3 is selected.

x : + : The feed is decelerated in the plus (+) direction.
- : The feed is decelerated in the minus (-) direction.

[Operation] When a signal becomes "0", the corresponding axis decelerates to stop in the specified direction.
If a specified feedrate is lower than the external deceleration rate, the specified feedrate is selected.

Manual handle feed maximum feedrate change signal HNDLF<Gn023.3>

[Classification] Input signal

[Function] This signal switches for the maximum handle feedrate.

[Operation] When this signal is 0, clamping is performed by the manual rapid traverse rate (parameter No. 1424) or parameter No. 1420. The external deceleration function is disabled for handle feed.
When this signal is 1, clamping is performed by the maximum handle feedrate (parameter No. 1434). The external deceleration function is enabled for handle feed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn118				*+ED5	*+ED4	*+ED3	*+ED2	*+ED1
Gn120				*-ED5	*-ED4	*-ED3	*-ED2	*-ED1
Gn101				*+ED25	*+ED24	*+ED23	*+ED22	*+ED21
Gn103				*-ED25	*-ED24	*-ED23	*-ED22	*-ED21
Gn107				*+ED35	*+ED34	*+ED33	*+ED32	*+ED31
Gn109				*-ED35	*-ED34	*-ED33	*-ED32	*-ED31
Gn023					HNDLF			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1005			EDMx	EDPx				

[Input type] Parameter input
[Data type] Bit axis

#4 EDPx In cutting feed, an external deceleration signal in the + direction for each axis is:
0: Invalid
1: Valid

> **NOTE**
> Be sure to set "1" to this parameter if bit 5 (EDR) of parameter No.1405 is set to 0 when positioning of linear interpolation type is used.

#5 EDMx In cutting feed, an external deceleration signal in the - direction for each axis is:
0: Invalid
1: Valid

> **NOTE**
> Be sure to set "1" to this parameter if bit 5 (EDR) of parameter No.1405 is set to 0 when positioning of linear interpolation type is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1405			EDR					

[Input type] Parameter input
[Data type] Bit path

#5 EDR As the external deceleration rate for positioning of linear interpolation type:
0: The external deceleration rate for cutting feed is used.
1: The external deceleration rate for the first axis in rapid traverse is used.
Let us use external deceleration 1 as an example.
When this parameter bit is set to 0, the value of parameter No. 1426 is used as the external deceleration rate for external deceleration 1.
When this parameter bit is set to 1, the value of axis 1 of parameter No. 1427 is used as the external deceleration rate for external deceleration 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1406							EX3	EX2

[Input type] Parameter input
[Data type] Bit path

#0 EX2 External deceleration function setting 2 is:
0: Invalid
1: Valid

#1 EX3 External deceleration function setting 3 is:
0: Invalid
1: Valid

1426	External deceleration rate of cutting feed

[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 an external deceleration rate for cutting feed or positioning of linear interpolation type (G00).

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.

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)
When manual handle feedrate switching signal HNDLF <Gn023.3> is 1, set a maximum manual handle feedrate for each axis.

1440	External deceleration rate setting 2 in cutting feed

[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 external deceleration rate 2 for cutting feed or positioning of linear interpolation type (G00).

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.

1442	Maximum manual handle feedrate setting 2 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 2 for each axis.

1443	External deceleration rate setting 3 in cutting feed

[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 external deceleration rate 3 for cutting feed or positioning of linear interpolation type (G00).

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.

	#7	#6	#5	#4	#3	#2	#1	#0
8131						EDC		

NOTE
When this parameter is set, the power must be turned off before operation is continued.

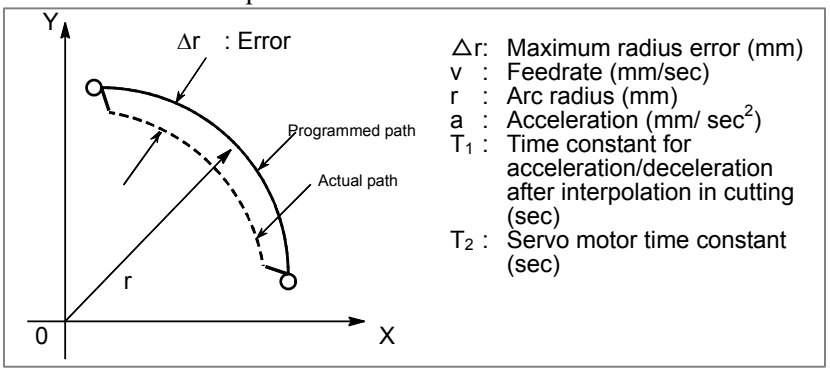
[Input type] Parameter input
[Data type] Bit

#2 EDC External deceleration is:
0: Not Used.
1: Used.

7.1.10 Acceleration-Based Speed Control in Circular Interpolation

Overview

When cutting is performed at high speed based on circular interpolation or helical interpolation, the actual tool path has an error relative to the programmed path. This error can be expressed approximately by the following equation in circular interpolation:



$$\Delta r = \frac{1}{2}(T_1^2 + T_2^2)\frac{v^2}{r} = \frac{1}{2}(T_1^2 + T_2^2)\cdot a \quad \cdots\cdots \text{(Equation 1)}$$

In actual machining, the allowable error Δr is given as the machining accuracy. Therefore, the allowable acceleration rate a (mm/sec^2) is determined by equation 1.
In acceleration-based speed control in circular interpolation, when a feedrate that causes a radius error to exceed the allowable error along a programmed arc with an arbitrary radius is specified, the feedrate for circular cutting is automatically clamped according to the parameter setting.
Let A be an allowable acceleration rate calculated from an allowable acceleration rate set for each axis. Then, the maximum allowable speed v for the programmed radius r is:

$$v = \sqrt{A \cdot r} \quad \cdots\cdots \text{(Equation 2)}$$

If a specified feedrate exceeds the speed v found from Equation 2, the feedrate is automatically clamped to v.

Parameter No. 1735 is used to specify an allowable acceleration rate. If allowable acceleration rates for two circular interpolation axes differ from each other, the smaller allowable acceleration rate is selected.
If the radius of an arc is small, the calculated reduced speed v may become very low. To prevent the feedrate from becoming too low in such a case, a minimum allowable feedrate can be set in parameter No. 1732.

Parameter

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.

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.

7.1.11 High Precision and High Speed Functions (Advanced Preview Control (T Series)/AI Advanced Preview Control (M Series)/AI Contour Control (M Series)/AI Contour Control II (M Series))

Overview

These functions are intended for high-speed, high-precision machining. The use of these functions suppresses the acceleration/deceleration delay that tends to increase as the feedrate becomes faster, as well as the delay in the servo system, reducing the machining profile error.

The high-speed, high-precision functions include:

(1) Advanced preview control (T series) = APC
(2) AI advanced preview control (M series) = AI APC
(3) AI contour control (M series) = AICC
(4) AI contour control II (M series) = AICC II

The following table shows the functions included in these functions.

	APC	AI APC		AICC	AICC II
Model	0*i*-TD	0*i* Mate-MD	0*i*-MD	0*i*-MD	0*i*-MD
Basic/option	Option	Basic		Option	Option
Look-ahead block count	1	12	20	40	200
Look-ahead linear acceleration/ deceleration before interpolation	○	○		○	○
Look-ahead bell-shaped acceleration/ deceleration before interpolation	—	—		☆	☆
Function for changing time constant of bell-shaped acceleration/deceleration	—	—		☆	☆
Advanced feed forward	○	○		○	○
Acceleration setting for each axis	○	○		○	○
Speed control based on the feedrate difference on each axis	○	○		○	○
Speed control with acceleration in circular interpolation	○	○		○	○
Speed control with the acceleration on each axis	—	○		○	○
Smooth speed control	—	—		—	○
Speed control with cutting load	—	—		—	○
Disregard of feedrate command	—	—		—	○
Jerk control **- Speed control with change of acceleration on each Axis** **- Look-ahead smooth bell-shaped acceleration/deceleration before Interpolation**	—	—		—	☆
Nano smoothing	—	—		—	☆

—: Function not supported
○: Standard function
☆: Optional function

The function for changing time constant of bell-shaped acceleration/deceleration is included in look-ahead bell-shaped acceleration/deceleration before interpolation.

Format

- Explanation

The high-speed, high-precision functions become available when you enter the advanced preview control (T series), AI advanced preview control (M series), AI contour control (M series) or AI contour control II (M series) mode.

T

- Advanced preview control

G08 P_ ;

P1 : Advanced preview control mode on
P0 : Advanced preview control mode off

NOTE
1 Always specify G08 in an independent block.
2 The advanced preview control mode is also cleared by the reset operation.

M

・AI advanced preview control / AI contour control / AI contour control II

G05.1 Q_ ;
Q1 : AI advanced preview control mode / AI contour control / AI contour control II mode on
Q0 : AI advanced preview control mode / AI contour control / AI contour control II mode off

NOTE
1 Always specify G05.1 in an independent block.
2 The AI advanced preview control / AI contour control / AI contour control II mode is also cleared by the reset operation.

Explanation

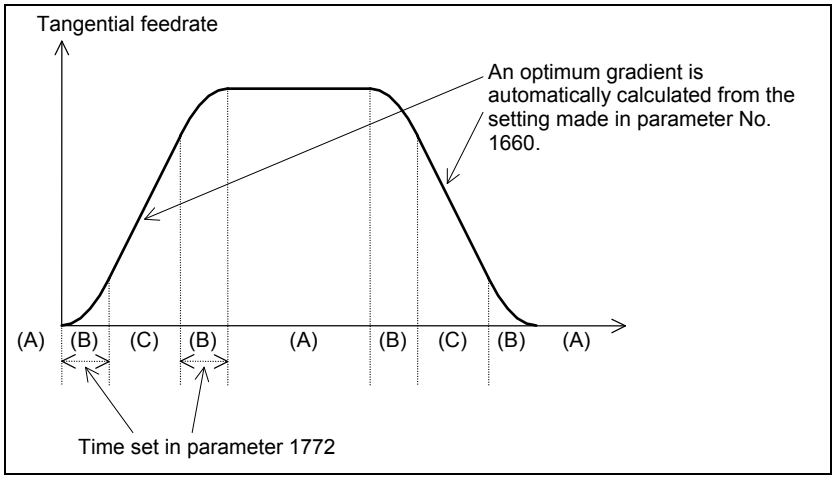
- Look-ahead acceleration/deceleration before interpolation

There are two look-ahead acceleration/deceleration before interpolation functions - look-ahead linear acceleration/deceleration before interpolation and look-ahead bell-shaped acceleration/deceleration before interpolation (M series). Look-ahead bell-shaped acceleration/deceleration before interpolation (M series) offers more smooth acceleration and deceleration.

* Look-ahead bell-shaped acceleration/deceleration before interpolation (M series) is an optional function.

- Setting an acceleration

A permissible acceleration for the linear acceleration/deceleration of each axis is set in parameter No. 1660. For bell-shaped acceleration/deceleration, acceleration change time (B) (period of transition from constant speed state (A) to constant acceleration/ deceleration state (C)) is set in parameter No. 1772. In the constant acceleration/deceleration state (C), acceleration/deceleration is performed with the maximum tangential acceleration not exceeding the permissible acceleration of each axis specified in parameter No. 1660.
The acceleration change time specified in parameter No. 1772 is held constant, regardless of the tangential acceleration.



- Method of determining the tangent acceleration

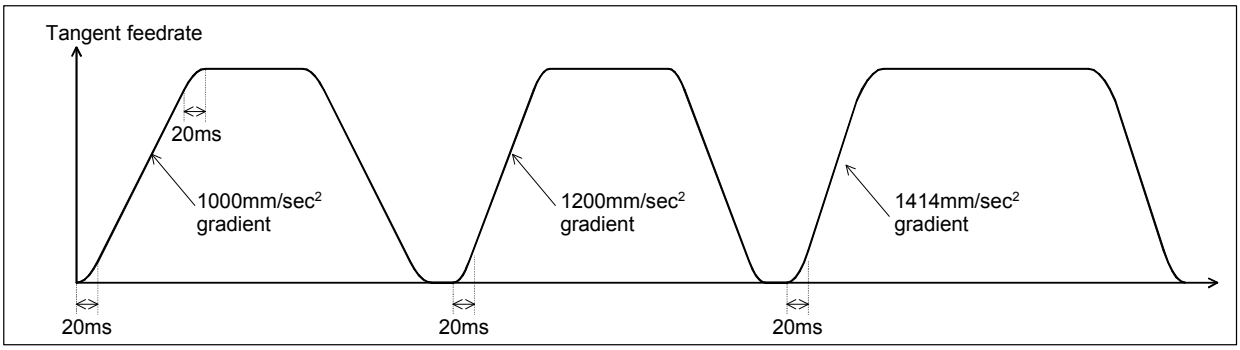
Acceleration/deceleration is performed with the largest tangent acceleration/deceleration that does not exceed the permissible acceleration set for each axis.

(Example)
X-axis permissible acceleration: 1000 mm/sec^2
Y-axis permissible acceleration: 1200 mm/sec^2
Acceleration change time: 20 msec

Program:

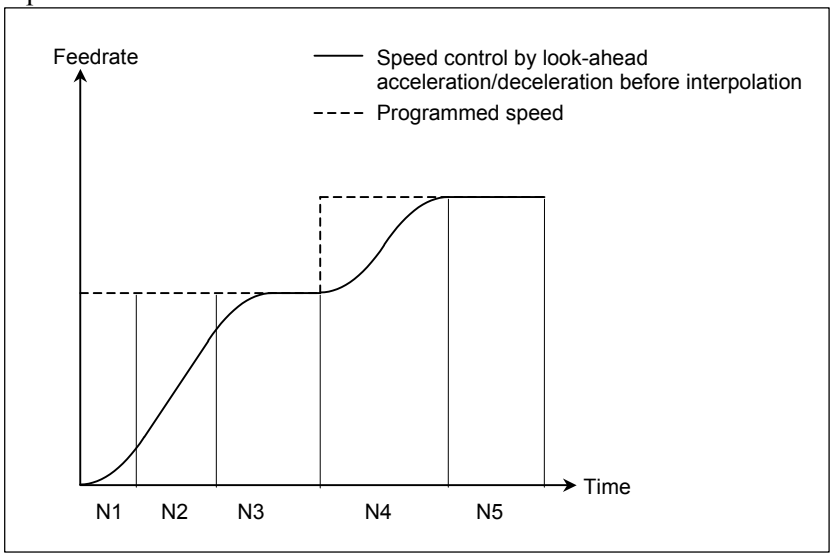
```
N1 G01 G91 X20. F6000 ; (Move on the X-axis.)
G04 X0.01 ;
N2 Y20. ; (Move on the Y-axis.)
G04 X0.01 ;
N3 X20. Y20. ; (Move in the XY direction (at 45 degrees).)
```

Since N3 performs interpolation for the X and Y axes in the 45-degree direction, the acceleration of the Y axis is controlled according to the X axis to become 1000 mm/s^2. Therefore, the combined acceleration is 1414 mm/s^2.



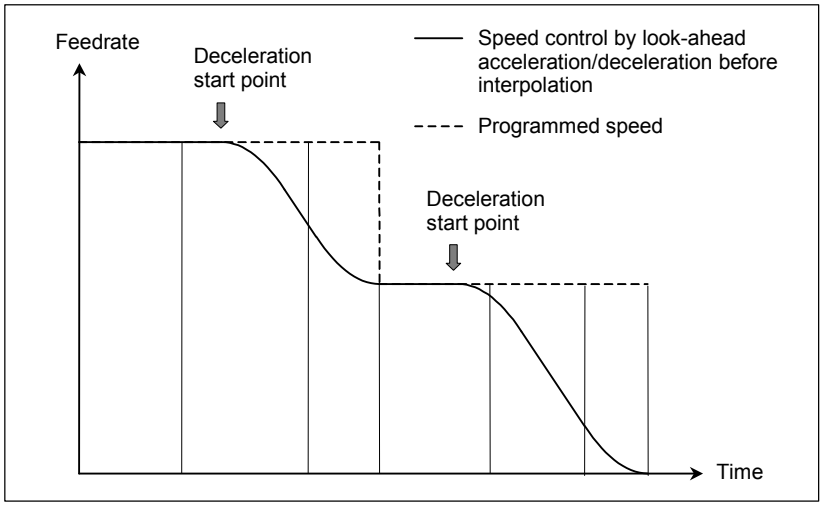
- Acceleration

Acceleration is performed so that the feedrate programmed for a block is attained at the beginning of the block. When look-ahead acceleration/deceleration before interpolation is valid for multiple blocks, acceleration can be performed across more than one block.



- Deceleration

Deceleration starts in advance so that the feedrate programmed for a block is attained at the beginning of the block.
When look-ahead acceleration/deceleration before interpolation is valid for multiple blocks, deceleration can be performed across more than one block.

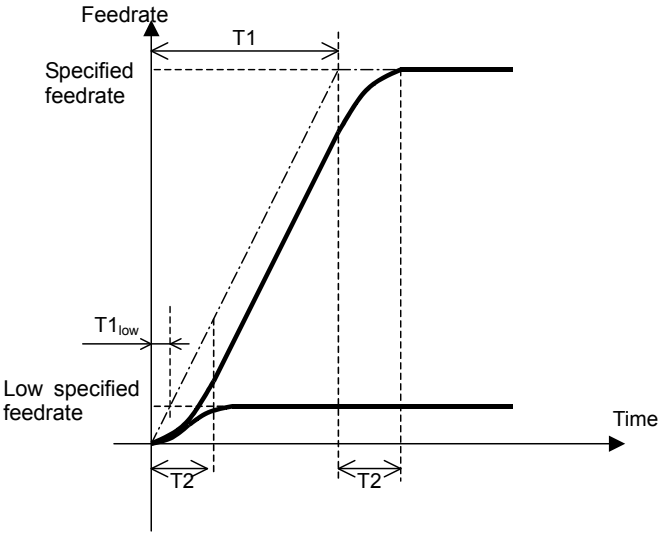


- Deceleration based on a distance

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.

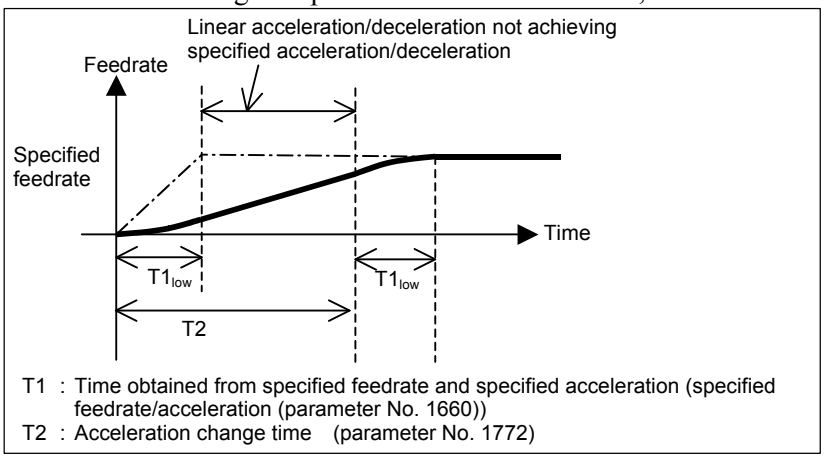
- Function for changing time constant of bell-shaped acceleration/deceleration

Bell-shaped acceleration/deceleration before interpolation is performed according to the acceleration and acceleration change time set by the parameters, as shown in the figure below.



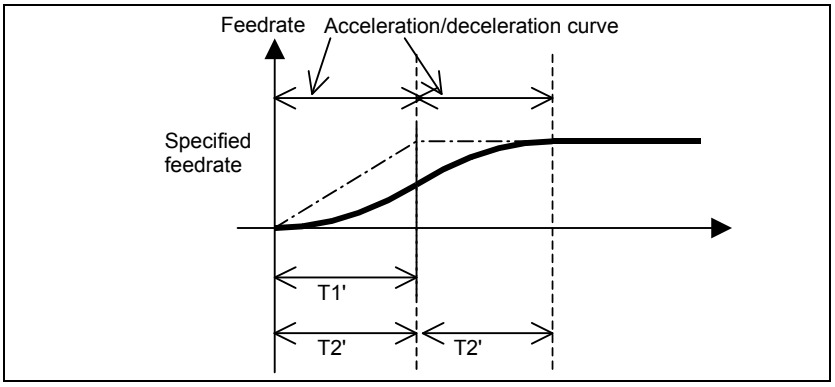
T1 : Time obtained from specified feedrate and specified acceleration (specified feedrate/acceleration (parameter No. 1660))
T2 : Acceleration change time (parameter No. 1772)

Here, the acceleration change time (T2) remains constant regardless of the specified feedrate, while the acceleration time for the linear section (T1), which is determined by acceleration, varies with the specified feedrate. If T1 becomes shorter than T2 when the specified feedrate is low, linear acceleration/deceleration not achieving the specified acceleration results, as shown in the figure below.



T1 : Time obtained from specified feedrate and specified acceleration (specified feedrate/acceleration (parameter No. 1660))
T2 : Acceleration change time (parameter No. 1772)

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.

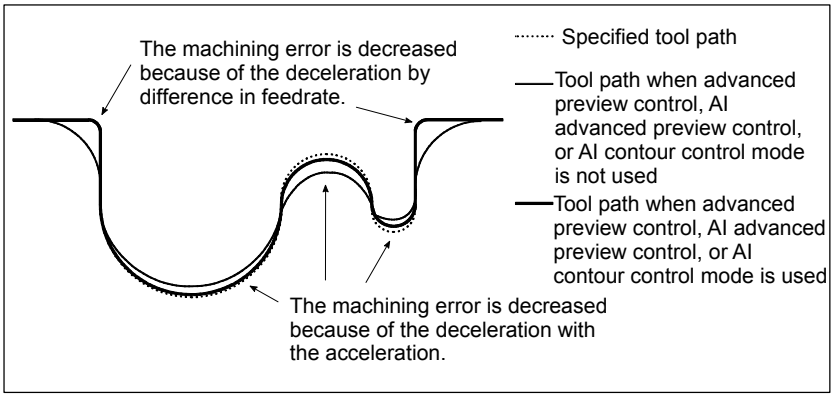
(1) Specifying the speed using an F in a G05.1 Q1 block
(2) Setting the speed on parameter No. 7066
(3) Setting the speed specified with the F command issued at the start of cutting as the reference speed

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

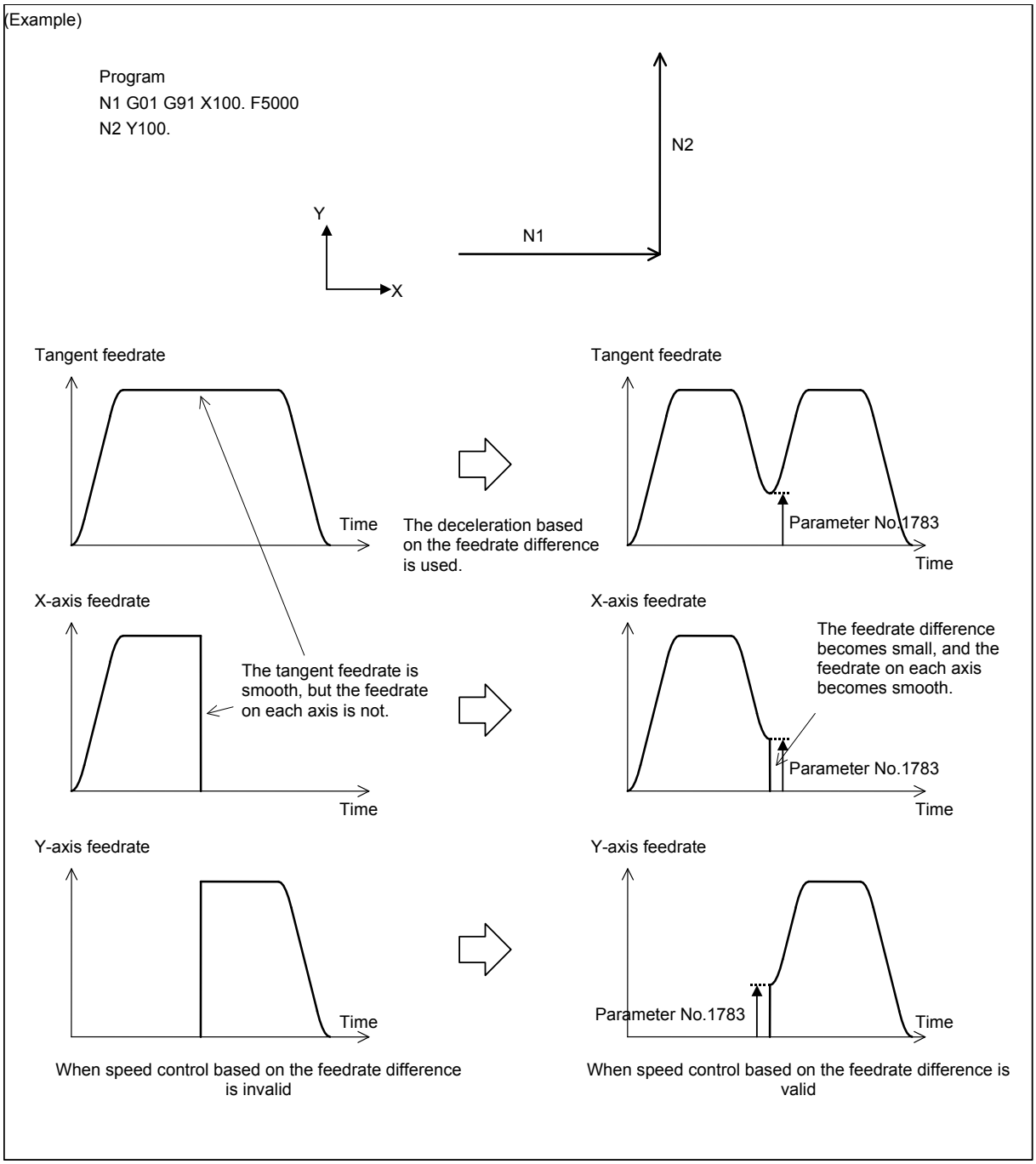
During the advanced preview control, AI advanced preview control, AI contour control, or AI contour control II mode, the feedrate is automatically controlled by reading blocks in advance.
The feedrate is determined using the following conditions. If the specified feedrate exceeds the determined feedrate, 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

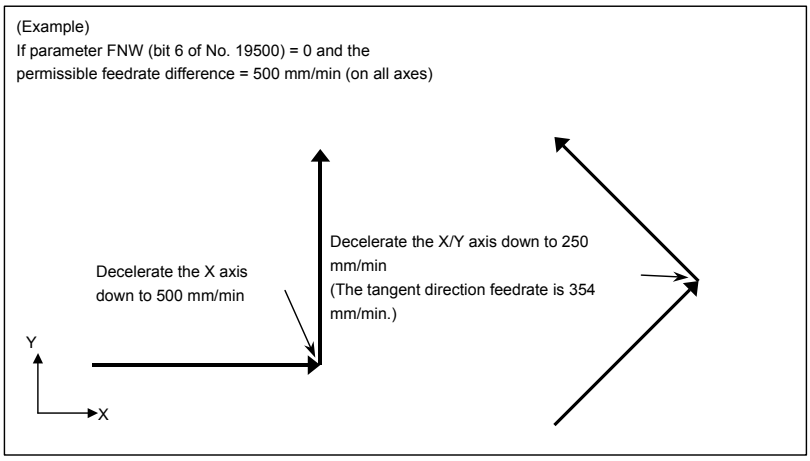


- Speed control based on the feedrate difference on each axis at a corner

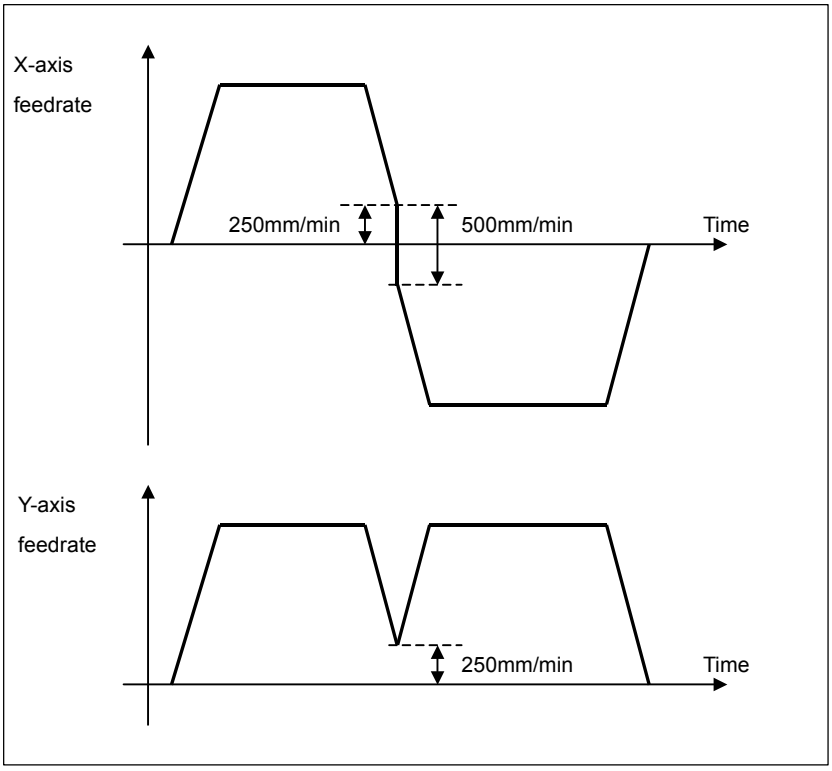
By using the speed control based on the feedrate difference on each axis at a corner, if a feedrate change occurs on an axis on each axis at a corner, the feedrate is determined so that any feedrate difference exceeding the permissible feedrate difference on that axis that has been set for parameter No. 1783 does not occur, and deceleration is automatically performed.



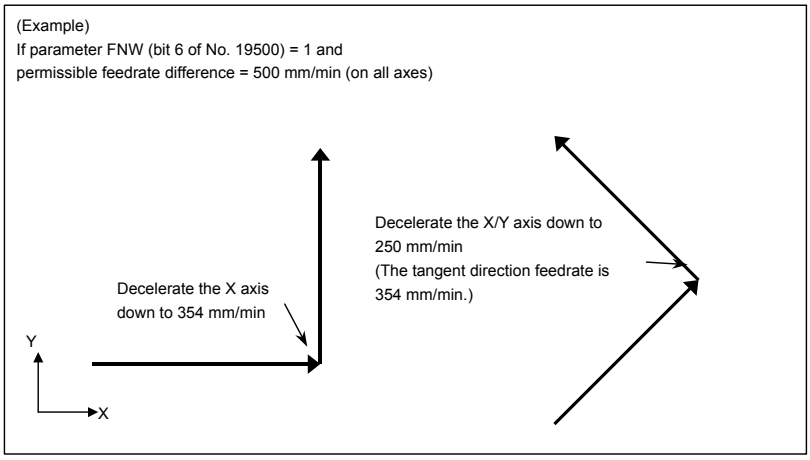
The method of deceleration based on the feedrate difference differs depending on the setting made for parameter FNW (bit 6 of No. 19500).
If "0" is set, the largest feedrate that does not exceed the permissible feedrate difference set for parameter No. 1783 is assumed to be the deceleration feedrate.
In this case, the deceleration feedrate differs if the travel direction differs, even if the shape is the same.



In the left-side example in the figure above, the X axis is inverted at the corner from the position direction to the negative direction, and deceleration is performed so that the feedrate difference becomes 500 mm/min. In other words, the feedrate is 250 mm/min both when the axis moves in the position direction and when it moves in the negative direction. As a result, the tangent direction feedrate becomes 354 mm/min.

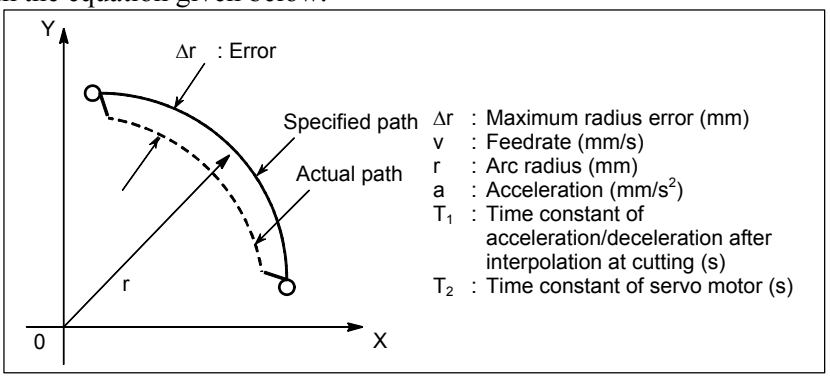


If "1" is set, the feedrate is determined not only with the condition that
the permissible feedrate difference and permissible acceleration on each axis are not exceeded, but also that the deceleration feedrate is constant regardless of the travel direction if the shape is the same.
If 1 is set for this 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 or helical 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, the permissible error Δr is given as the machining accuracy. Therefore, the permissible acceleration a (mm/sec^2) is determined by equation 1.
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:

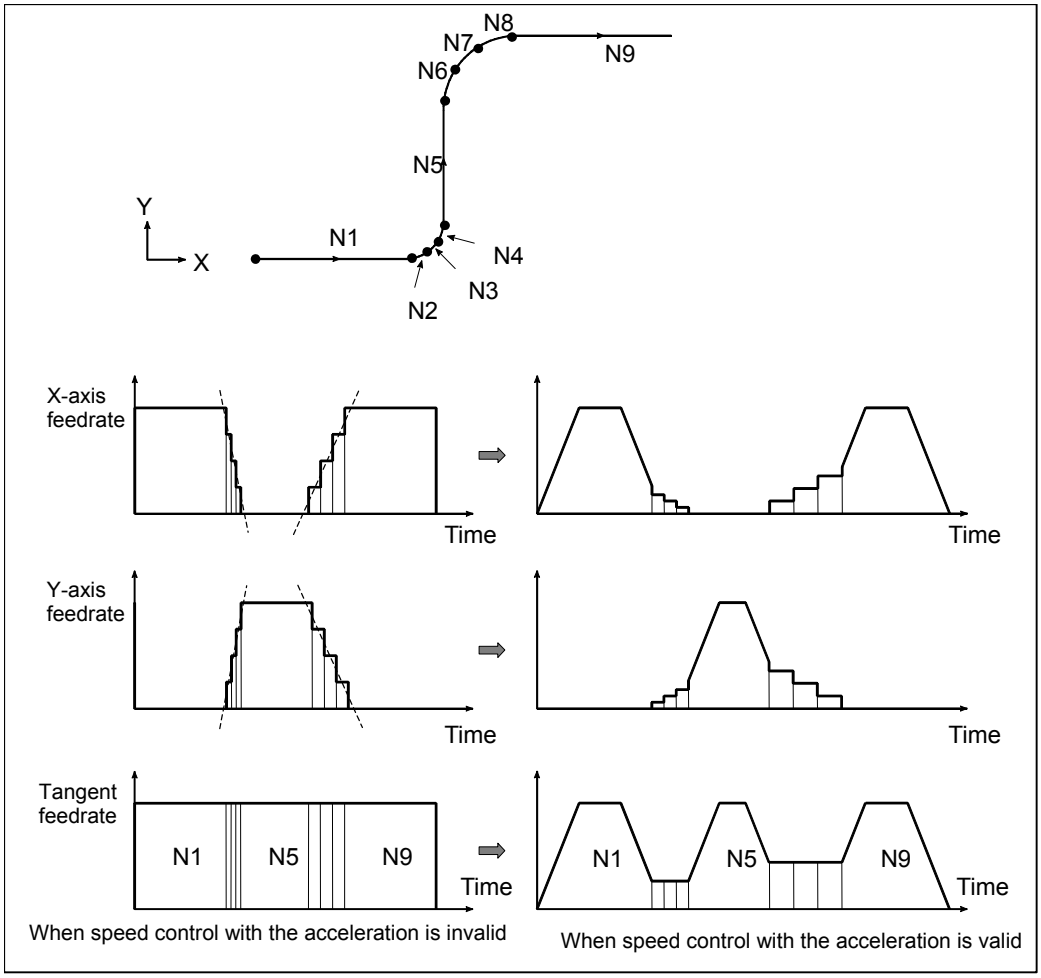
$$v = \sqrt{A \cdot r} \quad \text{.....(Equation 2)}$$

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.

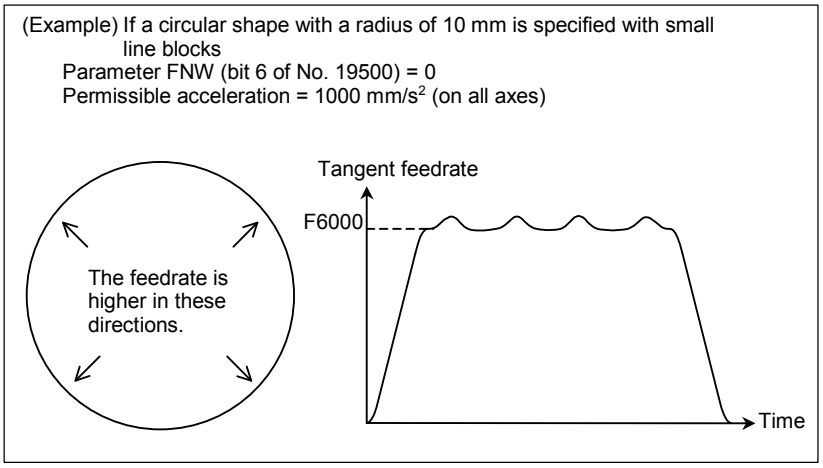
- Speed control with the acceleration on each axis

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.

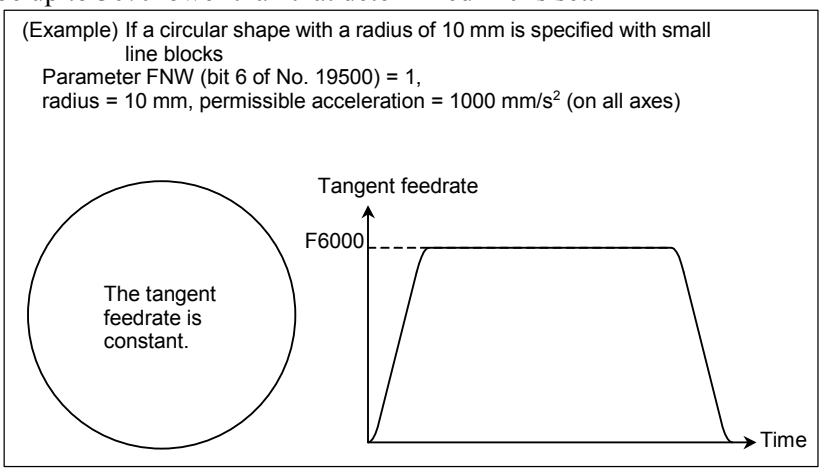
In the following example, the acceleration (gradient of the broken line in the feedrate graph) at too large at corners N2 to N4 and N6 to N8 and, therefore, deceleration is performed.



The method of determining the feedrate with the acceleration differs depending on the setting of parameter FNW (bit 6 of 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.

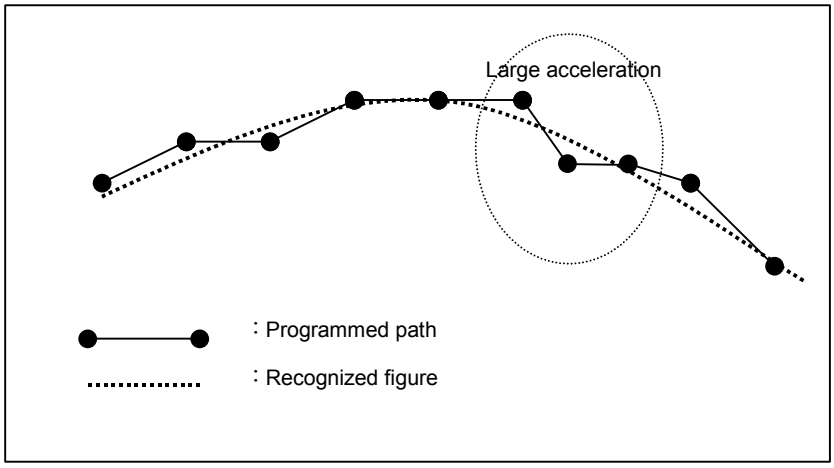


NOTE
In circular interpolation, the tangent feedrate is constant regardless of the setting of the parameter.

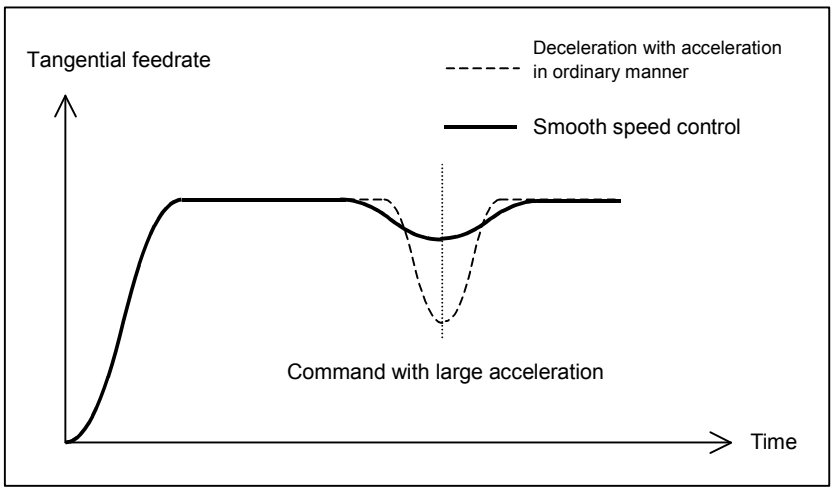
M

- 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 II 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.

M

- Deceleration 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 II, the tool travel direction on the Z-axis is used as a condition for calculating the machining feedrate.
This function is enabled when parameter ZAG (bit 4 of No. 8451) is set to 1.

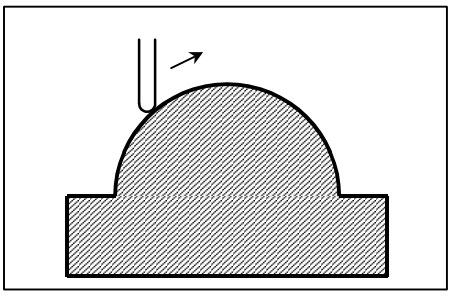


Fig. 7.1.11 (a) During ascent on the Z-axis

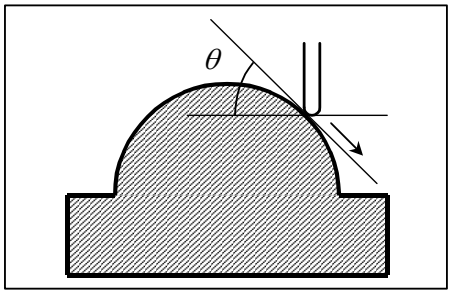


Fig. 7.1.11 (b) During descent on the Z-axis

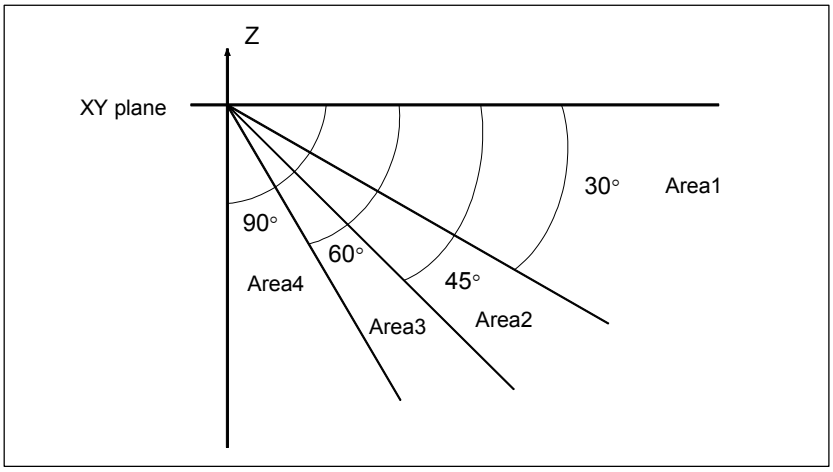
The descent angle θ during descent on the Z-axis (angle formed by the XY plane and the tool center path) is as shown in the Fig. 7.1.10 (b). The descent angle is classified into four areas, and the override values for the individual areas are set for the following parameters:

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 $30^\circ \leq \theta \leq 45^\circ$
Area3 $45^\circ \leq \theta \leq 60^\circ$
Area4 $60^\circ \leq \theta \leq 90^\circ$

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 system conversion, determine the descent angle on the Z-axis using the converted coordinate system.
4 Speed control with the cutting load is enabled for all interpolations in the AI contour control II mode. This function, however, can be made valid only for linear interpolations by setting bit 4 (ZOL) of parameter No. 19503 to 1.

M

- Ignoring feedrate commands

In a block in which AI contour control II is enabled, all feedrate commands (F commands) can be ignored by setting parameter NOF (bit 7 of 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 II is enabled
<2> F commands and modal F commands in the block in which AI contour control II 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 II 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 II.

- Another example of determining the feedrate

If a specified feedrate exceeds the upper feedrate limit of advanced preview control/AI advanced preview control/AI contour control/AI contour control II (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).

Limitations

- Conditions for temporarily canceling the advanced preview control, AI advanced preview control, AI contour control or AI contour control II mode

If any of the commands listed below is executed during one of these modes, the mode is temporarily canceled. Note that the advanced preview control, AI advanced preview control, AI contour control or AI contour control II mode resumes as soon as it becomes available.

Function name	G code	
	M series	T series
Positioning (rapid traverse) (NOTE 1)	G00	G00
Unidirectional positioning	G60	-
Spindle positioning	-	G00
Rigid tapping	G74, G84	G84, G88
Threading (NOTE 2)	G33	G32
Variable-lead threading (NOTE 2)	-	G34
Single threading cycle (NOTE 2)	-	G92
Multiple repetitive threading cycle (NOTE 2)	-	G76
Electronic box (EGB)	G81, G81.4	-
When no move command is specified	-	-
One-shot G code other than those shown at right (NOTE 1)	G09 G38, G39 G45, G46, G47, G48	G09 G38, G39

NOTE

1 If the first three conditions (1) to (3) below are all met, the mode is not canceled even when the rapid traverse command is specified. If all the conditions (1) to (5) are met, the mode is not canceled even when the G28, G30, or G53 command is specified.
   (1) Bit 1 (LRP) of parameter No. 1401 is set to 1. (Interpolation type positioning is valid.)
   (2) Parameter No. 1671 (maximum acceleration during rapid traverse) is set.
   (3) Bit 5 (FRP) of parameter No. 19501 is set to 1 (acceleration/deceleration before interpolation is valid for rapid traverse).
   (4) Bit 4 (ZRL) of parameter No. 1015 is set to 1 (the G28, G30, and G53 commands are of the interpolation type).
   (5) Bit 1 (AMP) of parameter No. 11240 is set to 1 (acceleration/deceleration before interpolation is valid for the G28, G30, and G53 commands in the high-speed, high-precision mode).
2 Acceleration/deceleration before interpolation is invalid for a threading command. Therefore, if acceleration/deceleration before interpolation is enabled by a command that precedes or follows a threading command, the tool is decelerated and stopped temporarily when the block changes.
   Since the state of acceleration/deceleration before interpolation does not change during continuous threading, deceleration does not occur when the block changes.

Signal

AI advanced preview control/AI contour control/AI contour control II mode on signal AICC <Fn062.0>

[Classification] Output signal

[Function] These signals indicate that the tool is running in the AI advanced preview control, AI contour control or AI contour control II mode.

[Output cond.] The signal is set to "1" when the condition for exerting AI advanced preview control, AI contour control or AI contour control II, such as a cutting command, is met when the AI advanced preview control, AI contour control or AI contour control II mode is on.
The signal is set to "0" in the following cases:

<1> Automatic operation is paused.
<2> Automatic operation is stopped.
<3> The condition for automatically canceling AI advanced preview control, AI contour control or AI contour control II, such as rapid traverse, is met.
<4> The AI advanced preview control, AI contour control or AI contour control II mode is canceled.

NOTE
If the first four conditions (1) to (4) below are all met, the output is done even when the rapid traverse command is specified. If all the conditions (1) to (6) are met, the output is done even when the G28, G30, or G53 command is specified.
(1) Bit 1 (AIR) of parameter No. 1612 is set to 1 (the status display and mode on signal are also valid during rapid traverse).
(2) Bit 1 (LRP) of parameter No. 1401 is set to 1. (Interpolation type positioning is valid.)
(3) Parameter No. 1671 (maximum acceleration during rapid traverse) is set.
(4) Bit 5 (FRP) of parameter No. 19501 is set to 1 (acceleration/deceleration before interpolation is valid for rapid traverse).
(5) Bit 4 (ZRL) of parameter No. 1015 is set to 1 (the G28, G30, and G53 commands are of the interpolation type).
(6) Bit 1 (AMP) of parameter No. 11240 is set to 1 (acceleration/deceleration before interpolation is valid for the G28, G30, and G53 commands in the high-speed, high-precision mode).

Advanced preview control mode on signal G08MD <Fn066.0>

[Classification] Output signal

[Function] This signal indicates that the advanced preview control mode is on.

[Output cond.] The signal is set to "1" when the condition for exerting advanced preview control, such as a cutting command, is met when the advanced preview control mode is on. The signal is set to "0" in the following cases:

<1> Automatic operation is paused.
<2> Automatic operation is stopped.
<3> The condition for automatically canceling advanced preview control is met.
<4> The advanced preview control mode is canceled.

NOTE

If the first four conditions (1) to (4) below are all met, the output is done even when the rapid traverse command is specified. If all the conditions (1) to (6) are met, the output is done even when the G28, G30, or G53 command is specified.

(1) Bit 1 (AIR) of parameter No. 1612 is set to 1 (the status display and mode on signal are also valid during rapid traverse).
(2) Bit 1 (LRP) of parameter No. 1401 is set to 1. (Interpolation type positioning is valid.)
(3) Parameter No. 1671 (maximum acceleration during rapid traverse) is set.
(4) Bit 5 (FRP) of parameter No. 19501 is set to 1 (acceleration/deceleration before interpolation is valid for rapid traverse).
(5) Bit 4 (ZRL) of parameter No. 1015 is set to 1 (the G28, G30, and G53 commands are of the interpolation type).
(6) Bit 1 (AMP) of parameter No. 11240 is set to 1 (acceleration/deceleration before interpolation is valid for the G28, G30, and G53 commands in the high-speed, high-precision mode).

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn062								AICC

	#7	#6	#5	#4	#3	#2	#1	#0
Fn066								G08MD

Parameter

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 advanced preview control, AI advanced preview control, AI contour control or AI contour control II. When the acceleration/deceleration before interpolation mode is not set, the maximum cutting feedrate set in parameter No. 1430 is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1602		LS2			BS2			

[Input type] Parameter input
[Data type] Bit path

#3 BS2 Acceleration/deceleration in a mode of acceleration/deceleration before look-ahead interpolation such as the advanced preview control, AI advanced preview control, AI contour control or AI contour control II mode:

0: Exponential acceleration/deceleration or linear acceleration/ deceleration is used. (The setting of bit 6 (LS2) of parameter No. 1602 is followed.)
1: Bell-shaped acceleration/deceleration is used.

#6 LS2 Acceleration/deceleration in a mode of acceleration/deceleration before interpolation such as the advanced preview control, AI advanced preview control, AI contour control or AI contour control II mode:

0: Exponential acceleration/deceleration is used.
1: Linear acceleration/deceleration is used.

BS2	LS2	Acceleration/deceleration
0	0	Exponential acceleration/deceleration after interpolation
0	1	Linear acceleration/deceleration after interpolation
1	0	Bell-shaped acceleration/deceleration after interpolation (The bell-shaped acceleration/deceleration after cutting feed interpolation option is required.)

	#7	#6	#5	#4	#3	#2	#1	#0
1611						AOF		

[Input type] Parameter input
[Data type] Bit path

#2 AOFF If the advanced preview feed forward function is enabled by parameter setting when the advanced preview control, AI advanced preview control, AI contour control or AI contour control II mode is not set, the advanced preview feed forward function is:

0: Enabled.
1: Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
1612							AIR	

[Input type] Parameter input
[Data type] Bit path

#1 AIR During rapid-traverse, the mode signals and status display in the advanced preview control, AI advanced preview control, AI contour control or AI contour control II mode are:

0: Invalid.
1: Valid.
When this parameter is set to 1, mode blink display, AI advanced preview control/AI contour control/AI contour control II mode signal AICC<Fn062.0> (M series), and advanced preview control signal G08MD<Fn066.0> (T series) are valid.

NOTE
In addition to the setting of this parameter, the following settings are required. In a rapid traverse command, the above items are valid when conditions 1 to 3 below are satisfied. In the G28, G30, and G53 commands, the above items are valid when conditions 1 to 5 below are satisfied.
1 Bit 1 (LRP) of parameter No. 1401 is 1 (interpolation type positioning is enabled).
2 Parameter No.1671 (maximum acceleration during rapid traverse) is set.
3 Bit 5 (FRP) of parameter No. 19501 is 1 (acceleration/deceleration before interpolation is enabled for rapid traverse).
4 Bit 4 (ZRL) of parameter No. 1015 is 1 (the G28, G30, and G53 commands are of interpolation type).
5 Bit 1 (AMP) of parameter No. 11240 is 1 (acceleration/deceleration before interpolation is enabled for the G28, G30, and G53 commands in the advanced preview control, AI advanced preview control, AI contour control or AI contour control II mode.)

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.

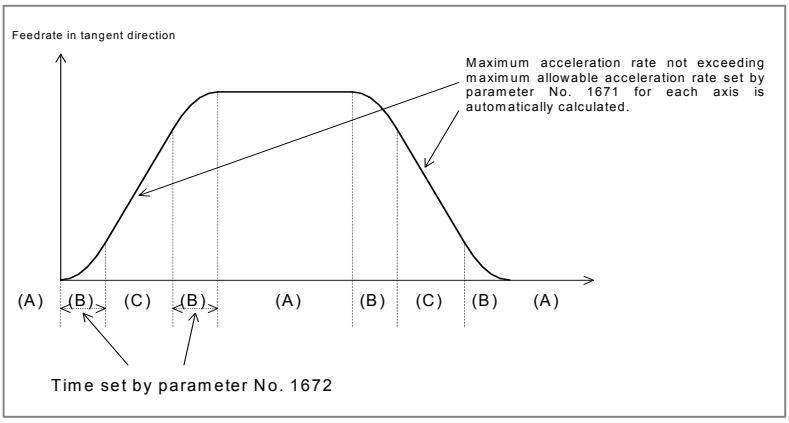
1671 **Maximum allowable acceleration rate in acceleration/deceleration before interpolation for linear rapid traverse for each axis**

[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 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.

1672	Acceleration change time of bell-shaped acceleration/deceleration before interpolation for linear rapid traverse

[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 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).



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.

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.

1737	
	Maximum allowable acceleration rate for the deceleration function based on acceleration in AI contour control/AI contour control II 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/AI contour control II (II)

[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 advanced preview control, AI contour control or AI contour control II, 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.

1763	
	FL rate for acceleration/deceleration after cutting feed interpolation for each axis in the acceleration/deceleration before interpolation mode

[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

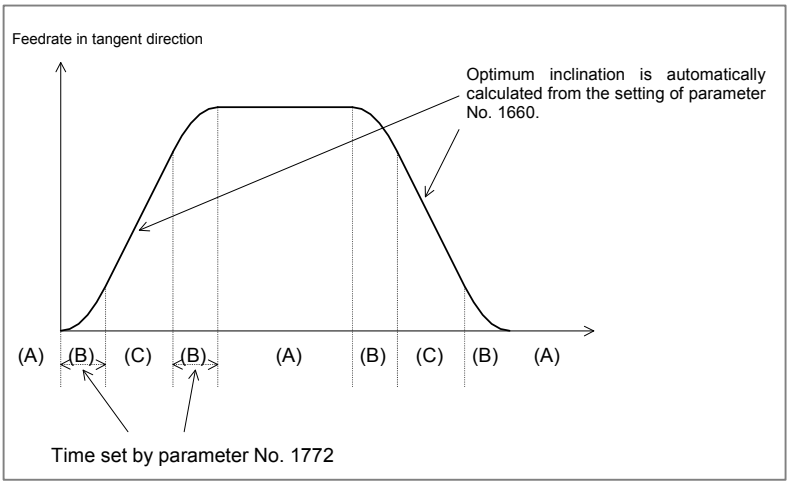
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +999000.0)
Set a minimum allowable feedrate (FL feedrate) for acceleration/ deceleration after cutting feed interpolation in acceleration/ deceleration before interpolation as in advanced preview control, AI advanced preview control, AI contour control or AI contour control II.

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 advanced preview control, AI advanced preview control, AI contour control or AI contour control II, 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 below).



NOTE
The option of bell-shaped acceleration/deceleration before look-ahead interpolation is required. This parameter is valid only in the AI contour control/AI contour control II mode.

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.

3241	Character blinking in the AI advanced preview control/AI contour control/AI contour control II mode (first character)
to	to
3247	Character blinking in the AI advanced preview control/AI contour control/AI contour control II mode (seventh character)

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0, 32 to 95
Set the first to seventh blinking characters in the AI advanced preview control/AI contour control/AI contour control II mode by using ASCII codes represented as decimal numbers.
When all parameters are set to 0, "AI APC" blinks for AI advanced preview control or "AICC" blinks for AI contour control or "AICC 2" blinks for AI contour control II.
Code numbers 032 to 095 in the "CHARACTER CODE LIST" can be set.

3251	Character blinking in the advanced preview control mode (first character)
to	to
3257	Character blinking in the advanced preview control mode (seventh character)

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0, 32 to 95
Set the first to seventh blinking characters in the advanced preview control mode by using ASCII codes represented as decimal numbers.
When 0 is set in all of these parameters, "APC" blinks.
Code numbers 032 to 095 in the "CHARACTER CODE LIST" can be set.

	#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 in AI contour control/AI contour control II mode 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 sets the acceleration/deceleration reference speed of the time constant change function of the bell-shaped acceleration/deceleration before interpolation in AI contour control/AI contour control II.

	#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 II (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 II, 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 II

8457	Override for range 3 that is applied during deceleration according to the cutting load in AI contour control II

8458	Override for range 4 that is applied during deceleration according to the cutting load in AI contour control II

[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 II, 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 $60° \leq \theta \leq 90°$

	#7	#6	#5	#4	#3	#2	#1	#0
8459					OVR			

[Input type] Parameter input
[Data type] Bit path

#3 OVR In deceleration by speed difference in advance preview control/AI advance preview control/AI contour control/AI contour control II, an override is:
0: Disabled.
1: Enabled.
An override is generally enabled for a specified speed and advance preview control/AI advance preview control/AI contour control/AI contour control II is applied to the specified speed. When this parameter is set to 1, an override is applied to the speed managed by advance preview control/AI advance preview control/AI contour control II.

8465 Upper limit of the speed of advance preview control/AI advance preview control/AI contour control/AI contour control II

[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 upper limit of the speed of advance preview control/AI advance preview control/AI contour control/AI contour control II.
If a speed higher than this parameter is specified in the advance preview control/AI advance preview control/AI contour control/AI contour control II mode, clamping is performed at the speed specified by this parameter.
When this parameter is 0, clamping is not performed.

8466 Upper limit of the speed of advance preview control/AI advance preview control/AI contour control/AI contour control II (when a rotation axis is specified independently)

[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 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 upper limit of the speed of advance preview control/AI advance preview control/AI contour control/AI contour control II during independent specification of a rotation axis.
If a speed higher that this parameter is specified in the advance preview control/AI advance preview control/AI contour control/AI contour control II mode, clamping is performed at the speed specified by this parameter.
When this parameter is 0, clamping is performed at the value specified by parameter No. 8465.

	#7	#6	#5	#4	#3	#2	#1	#0
11240							AMP	

[Input type] Parameter input
[Data type] Bit path

#1 **AMP** Movement from the midpoint to the reference position by the G28 or G30 command or movement by the G53 command in the advance preview control/AI advance preview control/AI contour control/AI contour control II mode uses:
0: Acceleration/deceleration after interpolation.
1: Acceleration/deceleration before interpolation.

NOTE
The setting of this parameter is valid when the following conditions are met.
1 Bit 4 (ZRL) of parameter No. 1015 is 1 (G28, G30, and G53 must be interpolation commands).
2 Bit 1 (LRP) of parameter No. 1401 is 1 (interpolation positioning is enabled).
3 Parameter No. 1671 (maximum acceleration during rapid traverse) is set.
4 Bit 5 (FRP) of parameter No. 19501 is 1 (acceleration/deceleration before interpolation is valid for rapid traverse).

To enable blinking display and signal output indicating the advance preview control/AI advance preview control/AI contour control/AI contour control II mode even when G28, G30, or G53 is specified, set bit 1 (AIR) of parameter No. 1612 to 1 in addition to the above settings.

	#7	#6	#5	#4	#3	#2	#1	#0
19500	FCC	FNW						

[Input type] Parameter input
[Data type] Bit path

#6 FNW In the speed determination method by the speed difference of advanced preview control/AI advance preview control/AI contour control/AI contour control II and the speed determination method by the acceleration of advanced preview control/AI contour control/AI contour control II:

0: The maximum speed that does not exceed the allowable speed difference or allowable acceleration is selected.

1: The feedrate is determined so that the allowable speed difference and allowable acceleration of each axis are not exceeded and that the deceleration speed is constant regardless of the movement direction if the shape is the same.

#7 FCC When there is an axis that requires one or more seconds for acceleration in acceleration/deceleration before look-ahead interpolation:

0: Emphasis is placed on precision, so that the specified feedrate may not be reached.

1: Emphasis is placed on speed, so that the specified feedrate is produced.

When this parameter is set to 1, the precision of curved interpolation such as circular interpolation may decrease.

	#7	#6	#5	#4	#3	#2	#1	#0
19501			FRP					

[Input type] Parameter input
[Data type] Bit path

#5 FRP linear-shaped rapid traverse in the advanced preview control/AI advance preview control/AI contour control/AI contour control II mode 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.

NOTE
The setting of this parameter is valid when the following conditions are met.
(1) Bit 1 (LRP) of parameter No. 1401 is 1 (interpolation type positioning is enabled).
(2) Parameter No.1671 (maximum acceleration during rapid traverse) is set.

To perform signal output and blinking display for indicating the advanced preview control/AI advanced preview control/AI contour control/AI contour control II mode, set bit 1 (AIR) of parameter No. 1612 to 1 in addition to the above settings.

	#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 II, smooth feedrate control is:
0: Not used.
1: Used.

#4 ZOL The deceleration function based on cutting load in AI contour control II (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 II (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 II

[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 II is used.
This parameter is valid only when bit 1 (ZG2) of parameter No. 19515 is set to 1.

- Parameter list

In the following table, the feature may be described by the following mark.

APC : Advanced preview control (T series)
AI APC : AI advanced preview control (M series)
AICC : AI contour control (M series)
AICC II : AI contour control II (M series)

Positioning

Parameter	Parameter No.			
	APC	AI APC	AICC	AICC II
Positioning type (non-linear (0)/interpolation (1))	1401#1 LRP			
Acceleration/deceleration type (acceleration constant (0)/time constant (1))	1603#4 PRT			
Acceleration/deceleration type (after interpolation (0)/before interpolation (1))	19501#5 FRP			
Time constant of acceleration/deceleration after interpolation in rapid traverse	1620			
Time constant of bell-shaped acceleration/deceleration after interpolation in rapid traverse	1621			

Parameter	Parameter No.			
	APC	AI APC	AICC	AICC II
Maximum permissible acceleration of acceleration/deceleration after interpolation in rapid traverse	1671			
Acceleration change time of bell-shaped acceleration/deceleration before interpolation in rapid traverse	1672			

Acceleration/deceleration before interpolation

Parameter	Parameter No.			
	APC	AI APC	AICC	AICC II
Maximum permissible acceleration of acceleration/deceleration before interpolation	1660			
Acceleration change time of bell-shaped acceleration/deceleration before interpolation	None		1772	
Valid/invalid state of the function for changing time constant of bell-shaped acceleration/deceleration before interpolation	None		7055#3 BCG	
Reference acceleration/deceleration speed for the function for changing time constant of bell-shaped acceleration/deceleration before interpolation	None		7066	

Acceleration/deceleration after interpolation

Parameter	Parameter No.			
	APC	AI APC	AICC	AICC II
Acceleration/deceleration type of acceleration/deceleration after interpolation in cutting feed	1602#3 BS2, 1602#6 LS2			
FL rate of acceleration/deceleration after interpolation in cutting feed	1763			
Time constant of acceleration/deceleration after interpolation in cutting feed	1769			

Speed control based on the feedrate difference on each axis

Parameter	Parameter No.			
	APC	AI APC	AICC	AICC II
Permissible feedrate difference when determining the feedrate based on the feedrate difference at a corner	1783			
Method of determining the feedrate based on the feedrate difference or based on the acceleration	19500#6 FNW			

Speed control with acceleration in circular interpolation

Parameter	Parameter No.			
	APC	AI APC	AICC	AICC II
Lower-limit feedrate for the deceleration function with the acceleration in circular interpolation	1732			
Permissible acceleration for the deceleration function with the acceleration in circular interpolation	1735			

Speed control with the acceleration on each axis

Parameter	Parameter No.			
	APC	AI APC	AICC	AICC II
Permissible acceleration for the deceleration function with the acceleration	None	1737		
Lower-limit feedrate for the deceleration function with the acceleration	None	1738		

Parameter	Parameter No.			
	APC	AI APC	AICC	AICC II
Method of determining the feedrate based on the feedrate difference or based on the acceleration	None	19500#6 FNW		

Others

Parameter	Parameter No.			
	APC	AI APC	AICC	AICC II
Maximum cutting feedrate during the mode of acceleration/deceleration before interpolation	1432			
Upper-limit feedrate for advanced preview control, AI advanced preview control, or AI contour control/AI contour control II	8465			
Upper-limit feedrate for advanced preview control, AI advanced preview control, or AI contour control/AI contour control II (when only the rotation axis is specified)	8466			

Alarms and messages

No.	Message	Description
0109	FORMAT ERROR IN G08	The value of P after G08 is other than 0 or 1 or is not specified.

・Status display

Depending on the control mode, one of the status indications shown below blinks at the lower right of the screen.

Control mode	Status display
Advanced preview control	APC
AI advanced preview control	AI APC
AI contour control	AICC
AI contour control II	AICC 2

- In the case of advanced preview control, the character string to be displayed can be changed using parameter Nos. 3251 to 3257.
- In the case of AI advanced preview control, AI contour control or AI contour control II, the character string to be displayed can be changed using parameter Nos. 3241 to 3247.

7.1.12 Speed Command Extension in Least Input Increment C

Overview

When the least input increment C (IS-C) is selected, the limitations indicated in Table 7.1.12 (a) and Table 7.1.12 (b) have conventionally been applied to the speed and acceleration parameters.

・Conventional speed and angular velocity parameters

Table 7.1.12 (a)

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min degree/min	IS-C	0.0001	0.0000 to +99999.9999
inch/min	IS-C	0.00001	0.00000 to +4000.00000

・Conventional acceleration and angular acceleration parameters

Table 7.1.12 (b)

Unit of data	Increment system	Minimum data unit	Valid data range
mm/sec^2 degree/sec^2	IS-C	0.0001	0.0000 to +99999.9999
inch/sec^2	IS-C	0.00001	0.00000 to +9999.99999

With an axis for which bit 7 (IESP) of parameter No. 1013 is set to 1 and IS-C is specified as the increment system, this function extends the parameter input limitations to those indicated in Table 7.1.12 (c) and Table 7.1.12 (d).
With this function, values greater than the conventionally allowed values can be set in the speed and acceleration parameters.
For example, the upper limit of the maximum rapid traverse rate and maximum cutting speed after this function is applied is:

- About 1000000 mm/min when the unit of data is mm, and IS-C is used

・Speed and angular velocity parameters when this function is used

Table 7.1.12 (c)

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min degree/min	IS-C	0.001	0.000 to +999000.000
inch/min	IS-C	0.0001	0.0000 to +9600.0000

・Acceleration and angular acceleration parameters when this function is used

Table 7.1.12 (d)

Unit of data	Increment system	Minimum data unit	Valid data range
mm/sec^2 degree/sec^2	IS-C	0.001	0.000 to +999999.999
inch/sec^2	IS-C	0.0001	0.0000 to +99999.9999

NOTE
Note that the least input increment and valid data range of set data vary from the conventional ones.

Limitation

- Time constant parameters for linear acceleration/deceleration after interpolation and bell-shaped acceleration/deceleration after interpolation

If linear acceleration/deceleration after interpolation or bell-shaped acceleration/deceleration after interpolation (M series) is used in an operation such as rapid traverse, cutting feed, or manual feed, the maximum allowable time constant value is a half of the conventional maximum allowable value.
The table below indicates the pertinent time constant parameters:

Parameter number	Meaning
1620	Time constant T or T1 used for linear acceleration/deceleration or bell-shaped acceleration/deceleration in rapid traverse for each axis
1621	Time constant T2 used for bell-shaped acceleration/deceleration in rapid traverse for each axis
1622	Time constant of acceleration/deceleration in cutting feed for each axis
1624	Time constant of acceleration/deceleration in jog feed for each axis
1626	Acceleration/deceleration time constant in threading cycles for each axis
1769	Time constant for acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode

Parameter number	Meaning
5271-5274	Time constant for acceleration/deceleration in rigid tapping extraction (first-stage to fourth-stage gear)
5365-5368	Bell-shaped acceleration/deceleration time constant in rigid tapping (M series) (first-stage to fourth-stage gear)

- VCMD waveform display function

As the speed increases, the amount of data that can be obtained for VCMD waveform display increases. So, the maximum treatable amount of data may be exceeded, resulting in incorrect waveform display.

- Positional deviation amount display on the servo setting screen

If the positional deviation amount exceeds -9999999 when a fast movement is made in the negative direction, the displayed number of digits decreases by one digit due to a limitation on display space.
To reference a correct positional deviation amount, check diagnosis number 300, "Servo positional deviation".

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1013	IESPx							

[Input type] Parameter input
[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#7 IESP When the least input increment is C (IS-C), the function to allow to set the larger value to the parameter of the speed and the acceleration:
0: Not used.
1: Used.

As for the axis which set this parameter when the least input increment is C (IS-C), the larger value can be set to the parameter of the speed and the acceleration.
The data ranges of the speed and acceleration parameters are as shown in Tables 7.1.12(c) and 7.1.12(d).
When this function is made effective, the digit number below the decimal point of the parameter on input screen is changed. The digit number below the decimal point decreases by one digit in case of the least input increment C (IS-C).

	#7	#6	#5	#4	#3	#2	#1	#0
2282					ISE64			

[Input type] Parameter input
[Data type] Bit axis

#3 ISE64 The speed limitation on the feed forward function (bit 1 (FEED) of parameter No. 2005 = 1) is:
0: Remains unchanged from the conventional value.
1: Extended.
When the feed forward function is enabled, the speed limitation on an axis for which this parameter is set is extended as indicated below if the increment system is IS-C.

	IS-C	
	Conventional	Extended
Speed [Unit: mm/min]	100000	999000

Notes

NOTE
1 To use this function, the corresponding servo software is required.
2 This function cannot be set for an axis under PMC axis control.

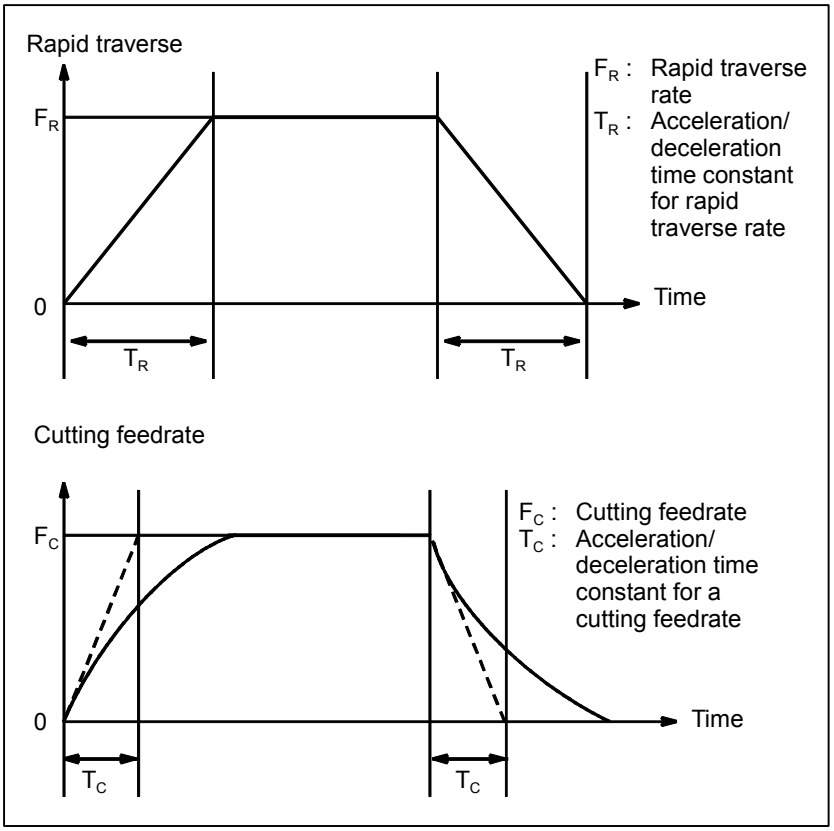
7.2 ACCELERATION/DECELERATION CONTROL

7.2.1 Automatic Acceleration/Deceleration

7.2.1.1 Automatic acceleration/deceleration

Overview

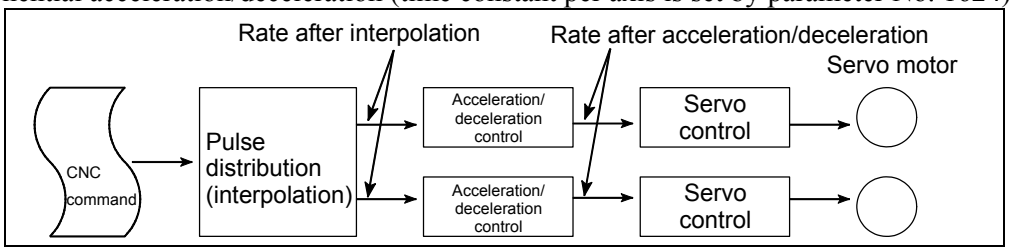
To prevent a mechanical shock, acceleration/deceleration is automatically applied when the tool starts and ends its movement (Fig. 7.2.1.1).



Acceleration and deceleration is performed when starting and ending movement, resulting in smooth start and stop.
Automatic acceleration/deceleration is also performed when feedrate changes, so the change in speed is also smooth.
It is not necessary to take acceleration/deceleration into consideration when programming.
Rapid traverse :
Linear acceleration/deceleration (time constant per axis is set by parameter No. 1620)
Cutting feed :
Exponential acceleration/deceleration (time constant per axis is set by parameter 1622)

Jog feed :

Exponential acceleration/deceleration (time constant per axis is set by parameter No. 1624)



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx			CTBx	CTLx

[Input type] Parameter input
[Data type] Bit axis

#0 CTLx Acceleration/deceleration in cutting feed or dry run

0: Exponential acceleration/deceleration is applied.
1: Linear acceleration/deceleration after interpolation is applied.

NOTE
When using bell-shaped acceleration/deceleration after interpolation, set this parameter to 0 and set bit 1 (CTBx) of parameter No. 1610 to select bell-shaped acceleration/deceleration after interpolation.

Parameter		Acceleration/deceleration
CTBx	CTLx	
0	0	Exponential acceleration/deceleration after interpolation
0	1	Linear acceleration/deceleration after interpolation
1	0	Bell-shaped acceleration/deceleration after interpolation

#1 CTBx Acceleration/deceleration in cutting feed or dry run

0: Exponential acceleration/deceleration or linear acceleration/ deceleration is applied. depending on the setting in CTLx, bit 0 of parameter No.1610)
1: Bell-shaped acceleration/deceleration is applied.

NOTE
This parameter is valid only when the bell-shaped acceleration/deceleration after cutting feed interpolation function is used. When this function is not used, the acceleration/deceleration is determined according to bit 0 (CTLx) of parameter No. 1610 regardless of the setting of this parameter.

#4 JGLx Acceleration/deceleration in jog feed

0: Exponential acceleration/deceleration is applied.
1: depending on which is used for cutting feed. (depending on the setting in CTBx or CTLx, bit 1 or 0 of parameter No.1610)

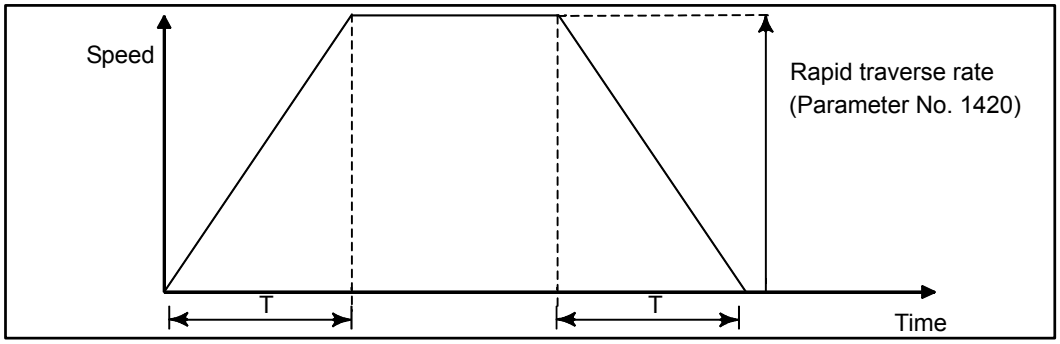
1620	Time constant T or T1 used for linear acceleration/deceleration or bell-shaped acceleration/deceleration in rapid traverse for each axis

[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.

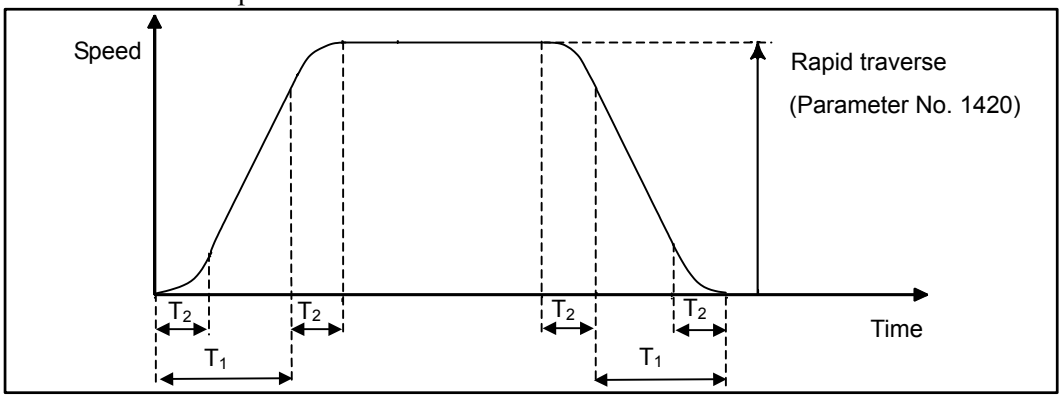
[Example]

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 : $T_2 \times 2$

1621	Time constant T2 used for 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 1000

Specify time constant T2 used for bell-shaped acceleration/ deceleration in rapid traverse for each axis.

1622	Time constant of exponential acceleration/deceleration or bell-shaped acceleration/deceleration after interpolation, or linear acceleration/deceleration after interpolation 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. 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.

1623	FL rate of exponential acceleration/deceleration after interpolation in cutting 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)
Set the lower limit (FL rate) of exponential acceleration/deceleration in cutting feed for each axis.

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.

1625	FL rate of exponential acceleration/deceleration in 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)
Set the FL rate of exponential acceleration/deceleration in cutting feed for each axis.
This parameter allows only the exponential type.

1626	Acceleration/deceleration time constant in threading cycles for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
Set a time constant for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis.

1627	FL rate for acceleration/deceleration in threading cycles 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 an FL feedrate for acceleration/deceleration after interpolation in the threading cycles G92 and G76 for each axis. Set 0 at all times except in a special case.

7.2.1.2 Rapid traverse block overlap

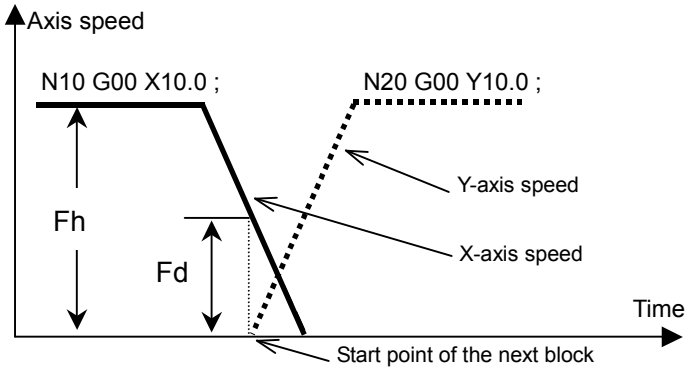
Overview

If rapid traverse blocks continue or the block next to a rapid traverse block does not move, the next block can be executed when the feedrate of each axis of a rapid traverse block is decreased to the reduction ratio set by the parameter.

Explanation

(Example)
N10 G00 X10.0 ;
N20 G00 Y10.0 ;

Fh: Rapid traverse rate
α: Reduction ratio (set by parameter No. 1722)
Fd: Deceleration evaluation rate = Fd×α/100



NOTE
Whether the speed is reduced to the reduction ratio is determined based on parameter No. 1420 (rapid traverse rate for each axis).

Limitation

Rapid traverse block overlap is disabled when:

- Pre-interpolation acceleration/deceleration is used for rapid traverse.
- The rapid traverse block overlap disable signal ROVLP <Gn053.3> is "1".
- In the rapid traverse command, dry run is enabled (bit 6 of parameter No. 1401 = 1) and DRN <Gn046.7> is "1". (This function is not disabled, however, if RT <Gn019.7> is set to "1".)
- Rapid traverse is of linear interpolation type (bit 1 of parameter No. 1401 = 1) and the acceleration/deceleration is constant over time (bit 4 of parameter No.1603 = 1).

Signal

Rapid traverse block overlap disable signal ROVLP<Gn053.5>

[Classification] Input signal
[Function] This signal disables rapid traverse block overlap.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn053			ROVLP					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601				RTO				

[Input type] Parameter input
[Data type] Bit path

#4 RTO Block overlap in rapid traverse
0: Blocks are not overlapped in rapid traverse.
1: Blocks are overlapped in rapid traverse.

1722	Rapid traverse feedrate reduction ratio for overlapping rapid traverse blocks

[Input type] Parameter input
[Data type] Byte axis
[Unit of data] %
[Valid data range] 0 to 100

This parameter is used when rapid traverse blocks are arranged successively, or when a rapid traverse block is followed by a block that does not cause, movement. When the feedrate for each axis of a block is reduced to the ratio set in this parameter, the execution of the next block is started.

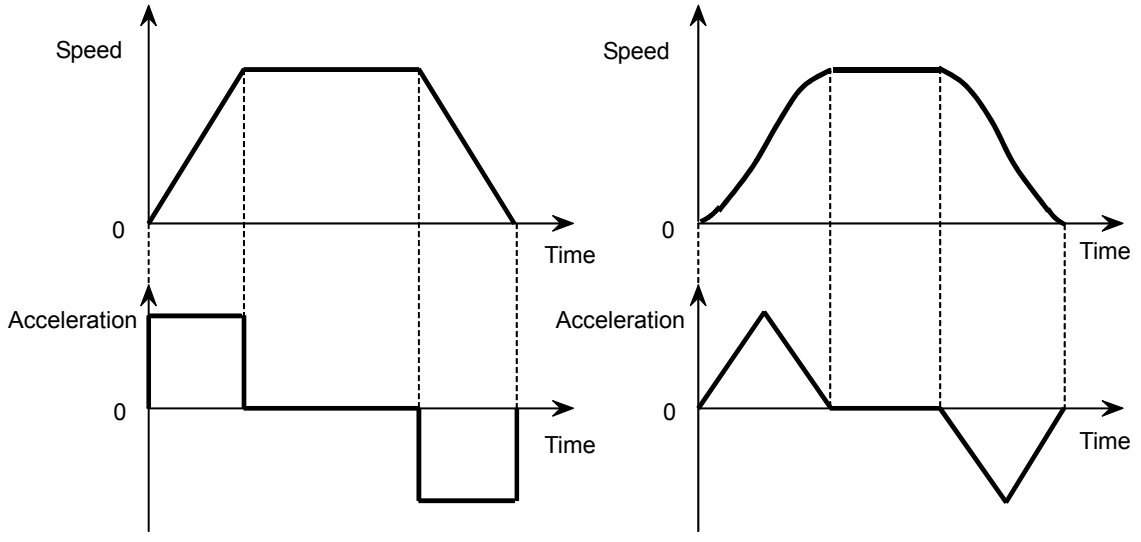
NOTE
The parameter No.1722 is effective when parameter No.1601 #4 (RTO) is set to 1.

7.2.2 Rapid Traverse Bell-shaped Acceleration/Deceleration

Overview

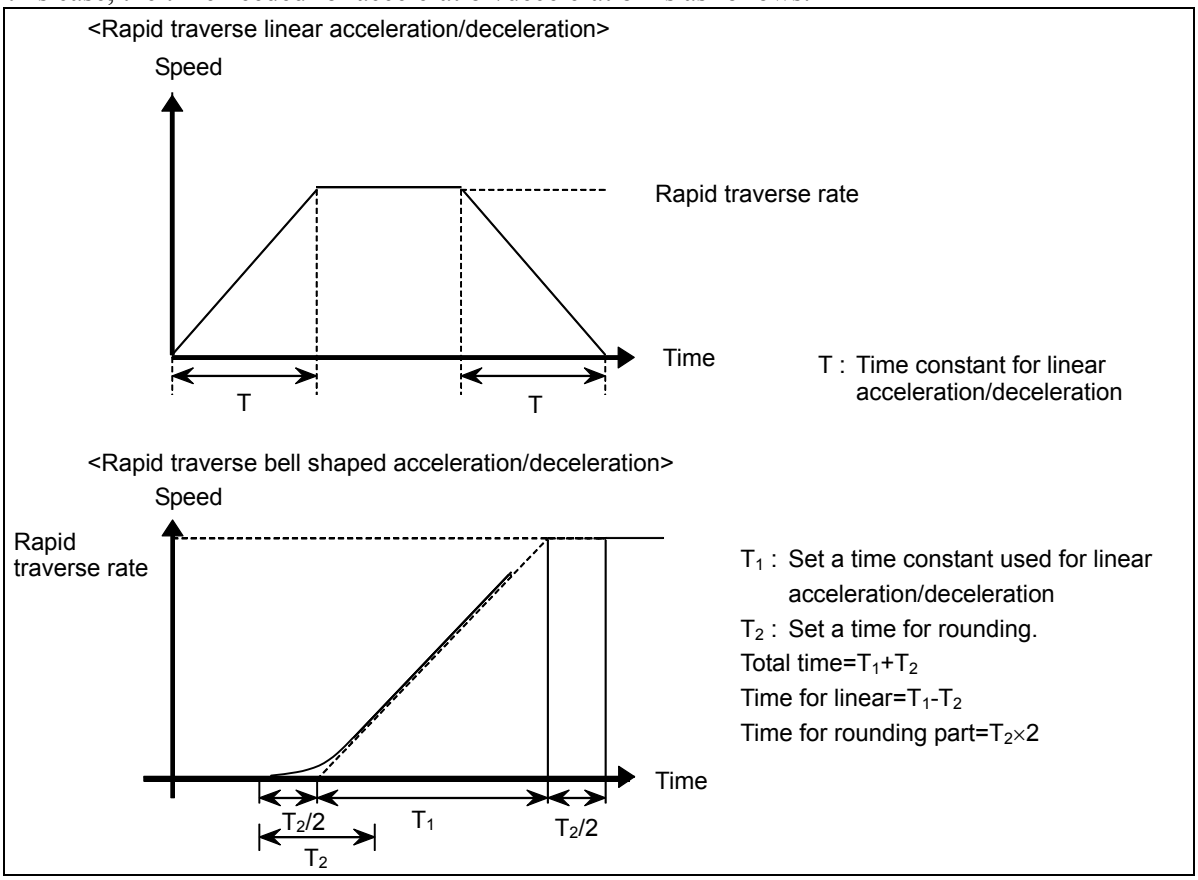
Rapid traverse bell-shaped acceleration/deceleration smoothly increases or decreases the rapid traverse rate, reducing the stress and strain imposed on the machine due to the variation in the acceleration with changes in the feedrate. As the time constant for bell-shaped acceleration/deceleration can be smaller than that for linear acceleration/deceleration, the time needed for acceleration/ deceleration can be reduced.

<Rapid traverse linear acceleration/deceleration> <Rapid traverse bell-shaped acceleration/deceleration>



This function is enabled when the time constants for rapid traverse bell-shaped acceleration/deceleration T_1 and T_2 are specified in parameter Nos. 1620 and 1621, respectively.

In this case, the time needed for acceleration/deceleration is as follows.



Set a time when rapid traverse override is 100%. When it is less than 100%, the total time is reduced (constant acceleration method).

Value of T_1 is determined from motor torque. Set a value of T_2 to 24 msec or 32 msec.

Parameter

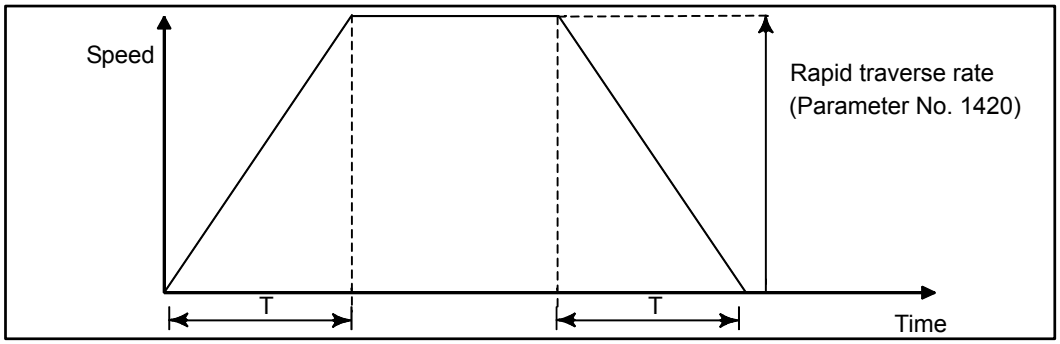
1620	**Time constant T or T1 used for linear acceleration/deceleration or bell-shaped acceleration/deceleration in rapid traverse for each axis**

[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.

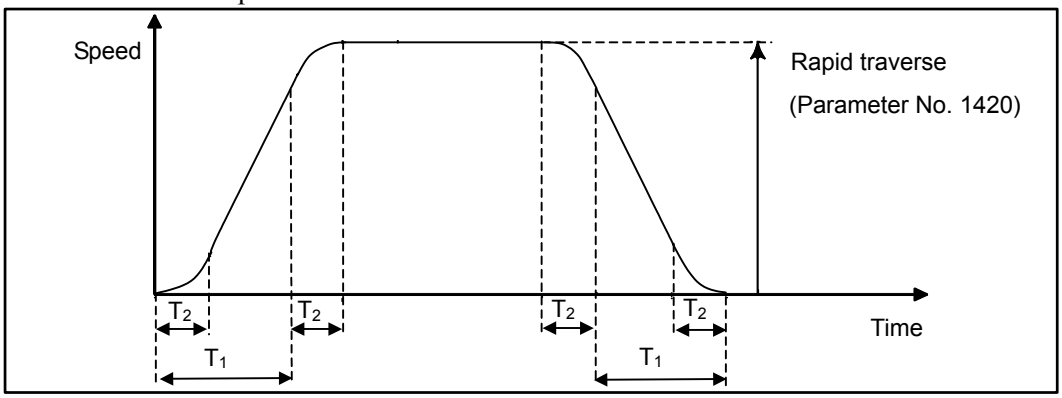
[Example]

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 : $T_2 \times 2$

1621	Time constant T2 used for 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 1000
Specify time constant T2 used for bell-shaped acceleration/ deceleration in rapid traverse for each axis.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (This manual)	Automatic acceleration/deceleration

7.2.3 Linear Acceleration/Deceleration after Cutting Feed Interpolation

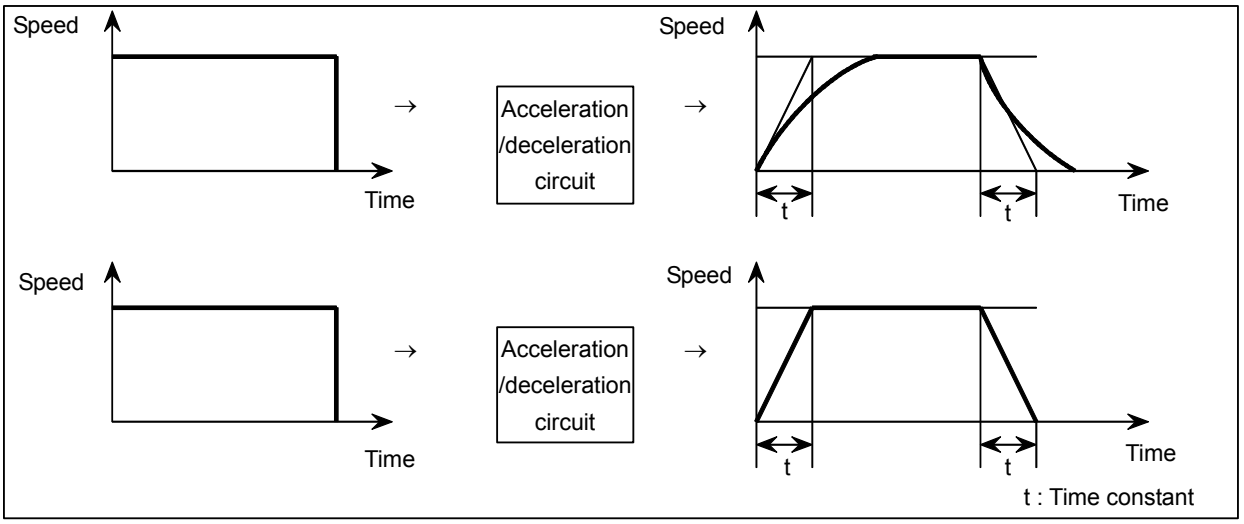
Overview

If linear acceleration/deceleration after interpolation for cutting feed is enabled (if bit 0 (CTL) of parameter No. 1610 is set to 1), acceleration/ deceleration is performed as follows:

Cutting feed:
Linear acceleration/deceleration (constant acceleration time)
Specify the acceleration/deceleration time constant for each axis in parameter No. 1622.

Jog feed:
Exponential or linear acceleration/deceleration (constant acceleration time)
Specify the acceleration/deceleration time constant for each axis in parameter No. 1624.

If an identical time constant is specified, linear acceleration/deceleration can halve the delay relative to the programmed time, in comparison with exponential acceleration/ deceleration, thus reducing the time needed for acceleration and deceleration. If circular interpolation is performed, especially when high-speed cutting is being performed, the actual tool path created after acceleration/deceleration will deviate from the programmed arc in the radial direction. This deviation can also be reduced, in comparison with exponential acceleration/deceleration, by applying linear acceleration/deceleration.



This function is enabled when bit 0 (CTL) of parameter No. 1610 is set to 1. If bell-shaped acceleration/deceleration after interpolation for cutting feed is also enabled, bell-shaped acceleration/deceleration is executed. The time constants for cutting feed and jog feed for each axis are specified in parameter Nos. 1622 and 1624 respectively, in the same way as for exponential acceleration/deceleration. The values specified for the FL feedrate for cutting feed (parameter No. 1623) and the FL feedrate for jog feed (parameter No. 1625) are ignored (always assumed to be 0).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx				CTLx

[Input type] Parameter input
[Data type] Bit axis

#0 CTLx Acceleration/deceleration in cutting feed or dry run
0: Exponential acceleration/deceleration is applied.
1: Linear acceleration/deceleration after interpolation is applied.

#4 JGLx Acceleration/deceleration in jog feed
0: Exponential acceleration/deceleration is applied.
1: Depending on which is used for cutting feed. (depending on the setting in CTBx or CTLx, bit 1 or 0 of parameter No.1610)

1622	Time constant of exponential acceleration/deceleration or bell-shaped acceleration/deceleration after interpolation, or linear acceleration/deceleration after interpolation 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. 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.

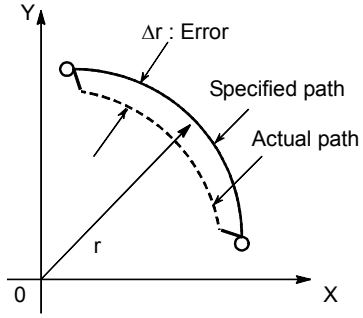
[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
Set the time constant used for acceleration/deceleration in jog feed for each axis.

Note

NOTE

(1) If linear acceleration/deceleration after interpolation for cutting feed is enabled, linear acceleration/deceleration is executed during cutting feed and during a dry run. Linear acceleration/deceleration can also be executed during jog feed if the parameter JGL (No. 1610#4) is specified accordingly.

(2) In circular interpolation especially when circular cutting is executed at high speed, the actual path of the accelerated or decelerated tool deviates from the specified arc in the direction of the radius.



Δr : Maximum radius error (mm)
v : Feedrate (mm/s)
r : Radius of arc (mm)
T_1 : Acceleration/deceleration time constant (sec)
T_2 : Time constant of servo loop (sec)
(Reciprocal of position loop gain)

The maximum error in the radial direction (Δr) can be approximated by the following expressions:

$$\Delta r = (\frac{1}{2}T_1^2 + \frac{1}{2}T_2^2)\frac{V^2}{r}$$ ..... Exponential acceleration/deceleration

$$\Delta r = (\frac{1}{24}T_1^2 + \frac{1}{2}T_2^2)\frac{V^2}{r}$$ .... Linear acceleration/deceleration or bell-shaped acceleration /deceleration after interpolation

If the error caused by the time constant of the servo loop is excluded, the error cause by linear acceleration/deceleration or bell shaped acceleration/deceleration after interpolation is 1/12 of that caused by exponential acceleration/deceleration.

(3) Linear acceleration/deceleration can be executed both for cutting feed and for jog feed along a PMC axis. Acceleration/deceleration for cutting feed is executed even if acceleration/deceleration for jog feed is selected. In jog feed along the PMC axis, the time constant for cutting feed is used instead of that for jog feed.

7.2.4 Bell-Shaped Acceleration/Deceleration after Cutting Feed Interpolation (M Series)

M

Overview

The bell-shaped acceleration/deceleration after cutting feed interpolation provides smooth acceleration and deceleration to reduce stress and strain on the machine. If this function is enabled (if bit 1 (CTB) of parameter No. 1610 is set to 1), acceleration/deceleration is performed as follows:

Cutting feed:

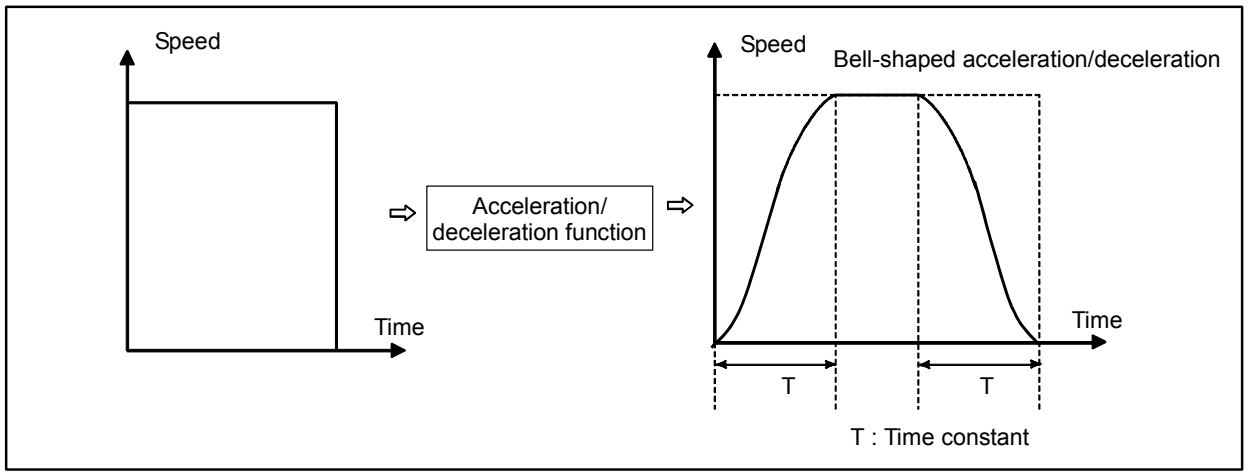
Bell-shaped acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter No. 1622.

Jog feed:

Exponential or bell-shaped acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter No. 1624.



Bell-shaped acceleration/deceleration after cutting feed interpolation is an optional function. This function is enabled when the CTB bit (bit 1 of parameter No. 1610) is specified. The time constants for cutting feed and for jog feed for each axis are specified in parameter Nos. 1622 and 1624 respectively, in the same way as exponential acceleration/deceleration. The values specified for the FL feedrate for cutting feed (parameter No. 1623) and the FL feedrate for jog feed (parameter No. 1625) are ignored (always assumed to be 0).

NOTE
Bell-shaped acceleration/deceleration function after cutting feed interpolation is optional function.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx			CTBx	CTLx

[Input type] Parameter input
[Data type] Bit axis

#0 CTLx Acceleration/deceleration in cutting feed or dry run
0: Exponential acceleration/deceleration is applied.
1: Linear acceleration/deceleration after interpolation is applied.

NOTE
When using bell-shaped acceleration/deceleration after interpolation, set this parameter to 0 and set bit 1 (CTBx) of parameter No. 1610 to select bell-shaped acceleration/deceleration after interpolation.

Parameter		Acceleration/deceleration
CTBx	CTLx	
0	0	Exponential acceleration/deceleration after interpolation
0	1	Linear acceleration/deceleration after interpolation
1	0	Bell-shaped acceleration/deceleration after interpolation

#1 CTBx Acceleration/deceleration in cutting feed or dry run
0: Exponential acceleration/deceleration or linear acceleration/ deceleration is applied. depending on the setting in CTLx, bit 0 of parameter No.1610)
1: Bell-shaped acceleration/deceleration is applied.

NOTE
This parameter is valid only when the bell-shaped acceleration/deceleration after cutting feed interpolation function is used. When this function is not used, the acceleration/deceleration is determined according to bit 0 (CTLx) of parameter No. 1610 regardless of the setting of this parameter.

#4 JGLx Acceleration/deceleration in jog feed
0: Exponential acceleration/deceleration is applied.
1: depending on which is used for cutting feed. (depending on the setting in CTBx or CTLx, bit 1 or 0 of parameter No.1610)

1622 **Time constant of exponential acceleration/deceleration or bell-shaped acceleration/deceleration after interpolation, or linear acceleration/deceleration after interpolation 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. 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.

Note

NOTE

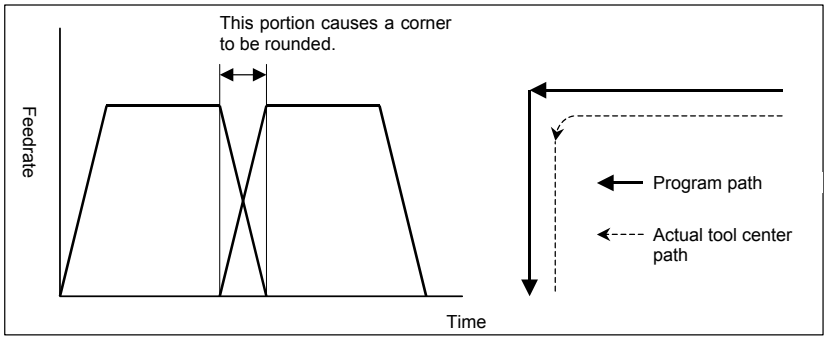
1 If bell-shaped acceleration/deceleration after interpolation during cutting feed is enabled, bell-shaped acceleration/deceleration is executed during cutting feed and during a dry run. Bell-shaped acceleration/deceleration can also be executed during jog feed if the parameter JGL (No. 1610#4) is specified accordingly.
2 In circular interpolation, the actual tool path after acceleration/deceleration deviates from the programmed arc in the radial direction. To overcome this radial deviation, see the note on linear acceleration/deceleration after interpolation for cutting feed.
3 Bell-shaped acceleration/deceleration can be executed both for cutting feed and for jog feed along a PMC axis. The time constant for acceleration/deceleration for jog feed is the same as that for cutting feed. In jog feed along the PMC axis, the time constant for cutting feed is used instead of that for jog feed.

7.2.5 Corner Control

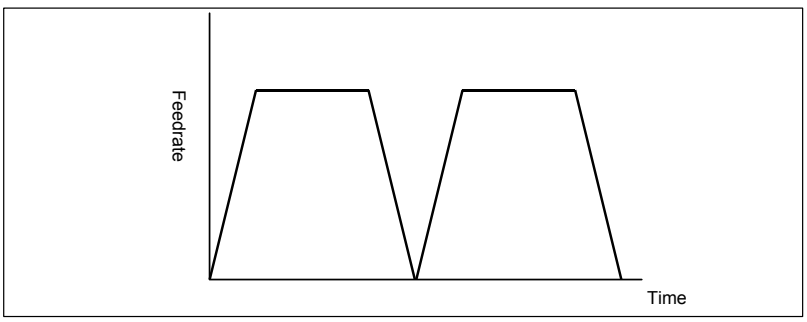
7.2.5.1 In-position check signal

Overvie

On general CNCs, the feedrate during cutting feed never becomes 0 between two successive blocks. So, a corner may be rounded.



By using the in-position check signal, control can be exercised so that machining can proceed to the next block after checking that the acceleration/deceleration processing of the previous block is completed and the effective area has been entered.



Signal

In-position check signal SMZ<Gn053.6>

[Classification] Input signal

[Function] This signal selects whether to make an in-position check between successive cutting feed blocks during automatic operation.
In general, slight rounding occurs between successive cutting feed blocks because of an acceleration/deceleration delay or servo delay.
By making an in-position check (to proceed to the next block after waiting until acceleration/deceleration and servo delays disappear), this rounding can be eliminated.

[Operation] When this signal is set to 0, machining proceeds to the next block without making an in-position check between successive cutting feed blocks during automatic operation.
When this signal is set to 1, machining proceeds to the next block after making an in-position check between successive cutting feed blocks during automatic operation.

NOTE
Before and after a block for positioning, an in-position check is made, regardless of the setting of this signal.
(However, the setting of bit 5 (NCI) of parameter No. 1601 is followed.)

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn053		SMZ						

7.2.5.2 In-position check

Overview

Whether the position of the servo motor is within a specified range is checked.
If the in-position check function is enabled, the CNC checks the position during deceleration. If the position is found to exceed the specified range, the CNC does not execute the next block.

NOTE
The purpose of in-position check is to check that the servo motor has reached within a specified range (specified with parameter No. 1826).

Signal

In-position signals INP1 to INP5 <Fn104.0 to Fn104.4>

[Classification] Output signal

[Function] These signals indicate that the control axes are set to the in-position condition.
They are provided for each control axis, and the end number in the signal name corresponds to the control axis number.
INPx
x : 1 ..... The 1st axis is set to the in-position condition.
2 ..... The 2nd axis is set to the in-position condition.
3 ..... The 3rd axis is set to the in-position condition.
: :

[Output cond.] These signals turn to "1" in the following case :
- The move command of the corresponding control axis is not specified and the servo error is within the specified allowance.

NOTE
Even if the servo error of the corresponding control axis is within the specified allowance (in-position width), the signal is not set to 1 when the axis is moving or there is acceleration/deceleration delay.

These signals turn to "0" in the following cases :

- The move command of the corresponding control axis is specified.
- The servo error of the corresponding control axis exceeds the specified allowance.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn104				INP5	INP4	INP3	INP2	INP1

Parameter

	#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.

1826	In-position width for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

The in-position width is set for each axis.
When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in-position width, the machine is assumed to have reached the specified position, or the machine is in the in-position state.

Note

NOTE

1 The in-position signals may turn to "1" even during the movement if the axis is fed at very low speed.
2 The in-position check function is enabled at the interface between two cutting blocks, when the exact stop command (G09) or exact stop mode command (G61) is specified

7.2.5.3 In-position check independently of feed/rapid traverse

Overview

If separate in-position check for cutting feed and rapid traverse is executed, a small in-position check range can be specified between those cutting feed blocks that require a high degree of precision. A large in-position check range can be specified between those rapid traverse blocks that require quick positioning.

Signal

See Item “In-position check.”.

Parameter

	#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. 1827).
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
1801			CIN	CCI				

[Input type] Parameter input
[Data type] Bit path

#4 CCI As the in-position width for cutting feed:
0: The parameter (No. 1826) applicable to rapid traverse as well is used.
1: The parameter (No. 1827) dedicated to cutting feed is used.
This parameter enables the in-position width for cutting feed (parameter No. 1827) to be set instead of the in-position width for rapid traverse (parameter No. 1826).
By setting bit 4 (CCI) of parameter No. 1801, choose whether to use this function or the conventional in-position check function.
This function, when specified, is enabled for all axes. So, for an axis that does not require this function, set the same data in parameter No. 1826 and No. 1827.

#5 CIN When CCI is set to 1, the dedicated parameter for specifying an in-position width for cutting feed is used:
0: Only when the next block specifies cutting feed.
1: Regardless of the next block.
The table below indicates the relationships between the parameters for cutting feed and rapid traverse.

		Parameter CIN(No.1801 #5)			
		0		1	
Parameter CCI (No.1801 #4)	0	Rapid traverse → Rapid traverse	No.1826	Rapid traverse → Rapid traverse	No.1826
		Rapid traverse → Cutting feed	No.1826	Rapid traverse → Cutting feed	No.1826
		Cutting feed → Cutting feed	No.1826	Cutting feed → Cutting feed	No.1826
		Cutting feed → Rapid traverse	No.1826	Cutting feed → Rapid traverse	No.1826
	1	Rapid traverse → Rapid traverse	No.1826	Rapid traverse → Rapid traverse	No.1826
		Rapid traverse → Cutting feed	No.1826	Rapid traverse → Cutting feed	No.1826
		Cutting feed → Cutting feed	No.1827	Cutting feed → Cutting feed	No.1827
		Cutting feed → Rapid traverse	No.1826	Cutting feed → Rapid traverse	No.1827

The parameters CCI and CIN can also be applied to a Cs axis.

1826 **In-position width for each axis**

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

The in-position width is set for each axis.
When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in-position width, the machine is assumed to have reached the specified position, or the machine is in the in-position state.

1827 **In-position width in cutting feed for each axis**

[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 each axis in cutting feed. This parameter is used when bit 4 (CCI) of parameter No.1801=1.

Note

NOTE
If the parameter NCI (No. 1601#5) is set to 1, so that position check is not performed during deceleration, this function is invalid. The system starts execution of the next block as soon as deceleration has been completed, without checking whether the servo position error is within the specified range.

7.2.6 Feed Forward in Rapid Traverse

Overview

Feed-forward control can be available even during rapid traverse. In this case, the servo position deviation is reduced, thus it reduces the time required for positioning to within the specified range.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1800					FFR			

[Input type] Parameter input
[Data type] Bit path

#3 FFR Feed-forward control in rapid traverse is:
0: Disabled
1: Enabled
Feed-forward is enabled only in normal cutting feed. But when this parameter is set to 1, feed-forward is enabled at rapid traverse as well. This capability reduces the servo positional deviation, thus it reduces the time required to enter the in-position width at the time of positioning.

Reference item

For details of this function, refer to the "FANUC SERVO MOTOR αi series Maintenance Manual (B-65285EN)."

7.3 JERK CONTROL (M Series)

M

7.3.1 Speed Control with Change of Acceleration on Each Axis

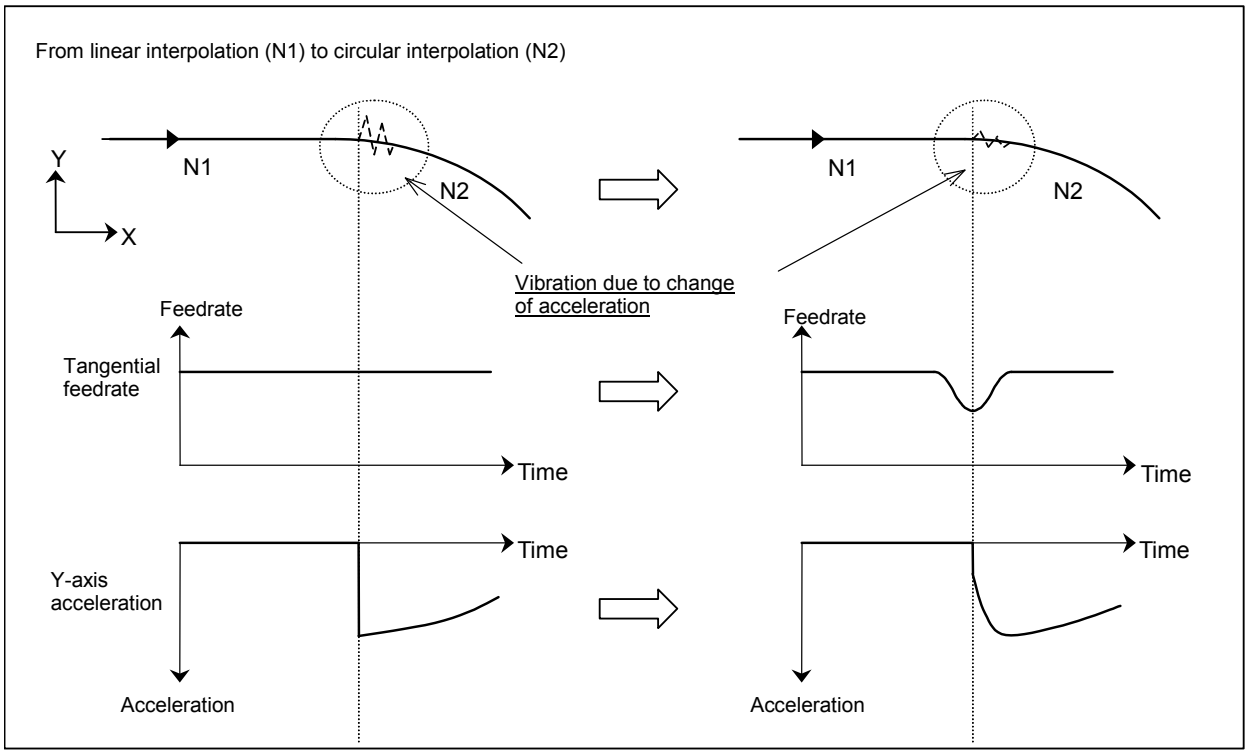
Overview

In portions in which acceleration changes largely, such as a portion where a programmed figure changes from a straight line to curve, vibration or shock on the machine may occur. Speed control with change of acceleration on each axis is a function to suppress machining errors due to vibration and machine shock generated by change of acceleration. This function obtains a feedrate so that change of acceleration is within the parameter-set permissible acceleration change amount for each axis, and performs deceleration by using acceleration/deceleration before interpolation.

CAUTION
Before speed control with change of acceleration on each axis can be used, the options for jerk control and AI contour control II are required.

Explanation

In the following example, the Y-axis acceleration changes largely at the contact point between a linear interpolation and circular interpolation, so deceleration is performed.



- Setting the permissible acceleration change amount

The permissible acceleration change amount for each axis is set in parameter No. 1788. When 0 is set in this parameter for a certain axis, speed control with change of acceleration is not performed for that axis.

- Parameter setting example

Suppose a figure shown Fig. 7.3.1 in which a straight line is followed by an arc. Let the specified feedrate and the arc radius be 6000 mm/min and 10 mm, respectively. Then, the Y-axis acceleration change amount at the contact point of the linear and arc portions is obtained as follows:

$$\frac{v^2}{r} = 1000 mm/s^2$$

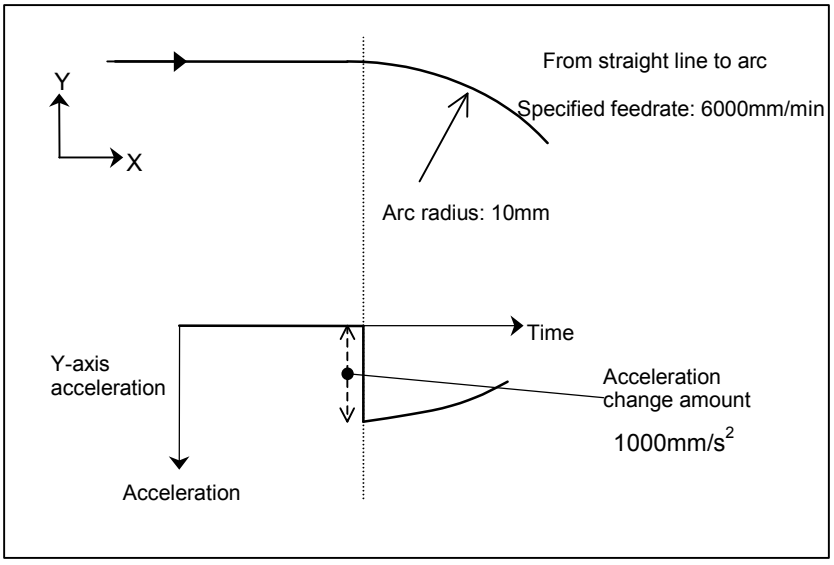
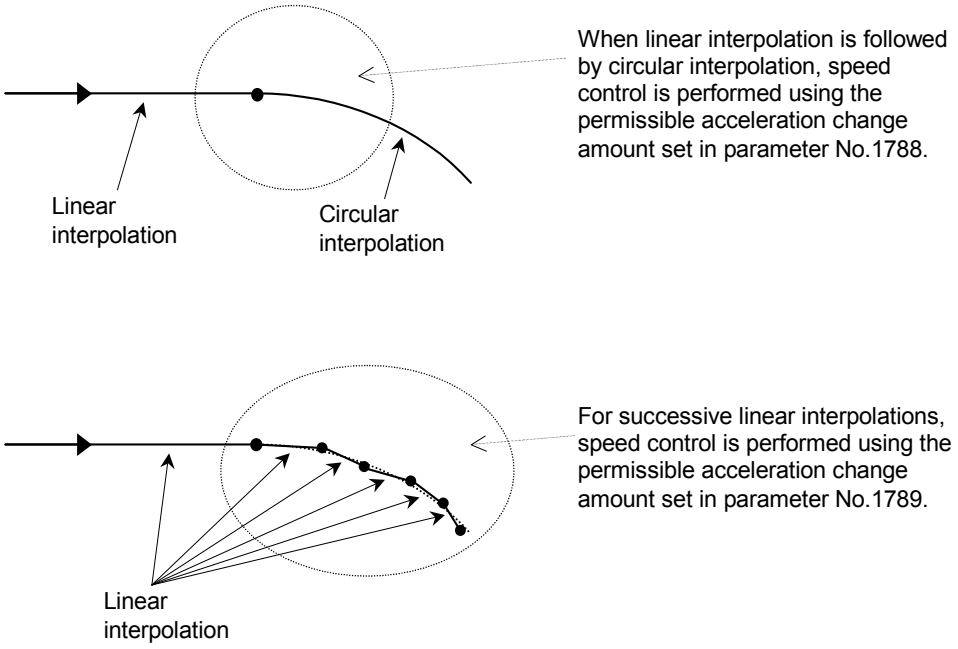


Fig. 7.3.1

To suppress the change of acceleration to 300 mm/sec^2, set 300 mm/sec^2 for the Y-axis in parameter No.1788.
Note that the change of acceleration is determined from the interpolation data of the CNC, so it may differ from the theoretical value.
The actual machine is affected by acceleration/deceleration and other factors, so the value to be set in the parameter should be determined after adjustments are made.

- For successive linear interpolations

When there are successive linear interpolations, speed control with change of acceleration obtains the deceleration feedrate from the change in acceleration between the start point and end point of a specified block.
When a curve is specified using 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. The error due to rounding may increase change of acceleration, and especially when the line segments specified by blocks are short, deceleration is performed frequently. As a result, the machining speed cannot increase enough. In such a case, a relatively large value should be set in parameter No. 1789 as the permissible acceleration change amount for each axis in successive linear interpolations to improve the machining speed.
When a value other than 0 is set in parameter No. 1789 for an axis for which deceleration with change of acceleration is enabled, this setting is regarded as the permissible acceleration change amount at corners in which linear interpolations meet. (For portions where a linear interpolation and circular interpolation meet and where circular interpolations meet, the setting in parameter No. 1788 is used.)
When 0 is set in parameter No. 1789 for an axis, the setting in parameter No. 1788 specifying the ordinary permissible acceleration change amount is used even at a corner in which linear interpolations meet.
When smooth speed control is used in speed control with permissible acceleration in AI contour control II, the deceleration feedrate is obtained from the change of acceleration calculated by smooth speed control. Therefore, the deceleration feedrate may be higher than the ordinary deceleration feedrate.



Parameter

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.

Note

NOTE
To use feedrate control based on an acceleration change of each axis, the acceleration control option and AI contour control II option are required.

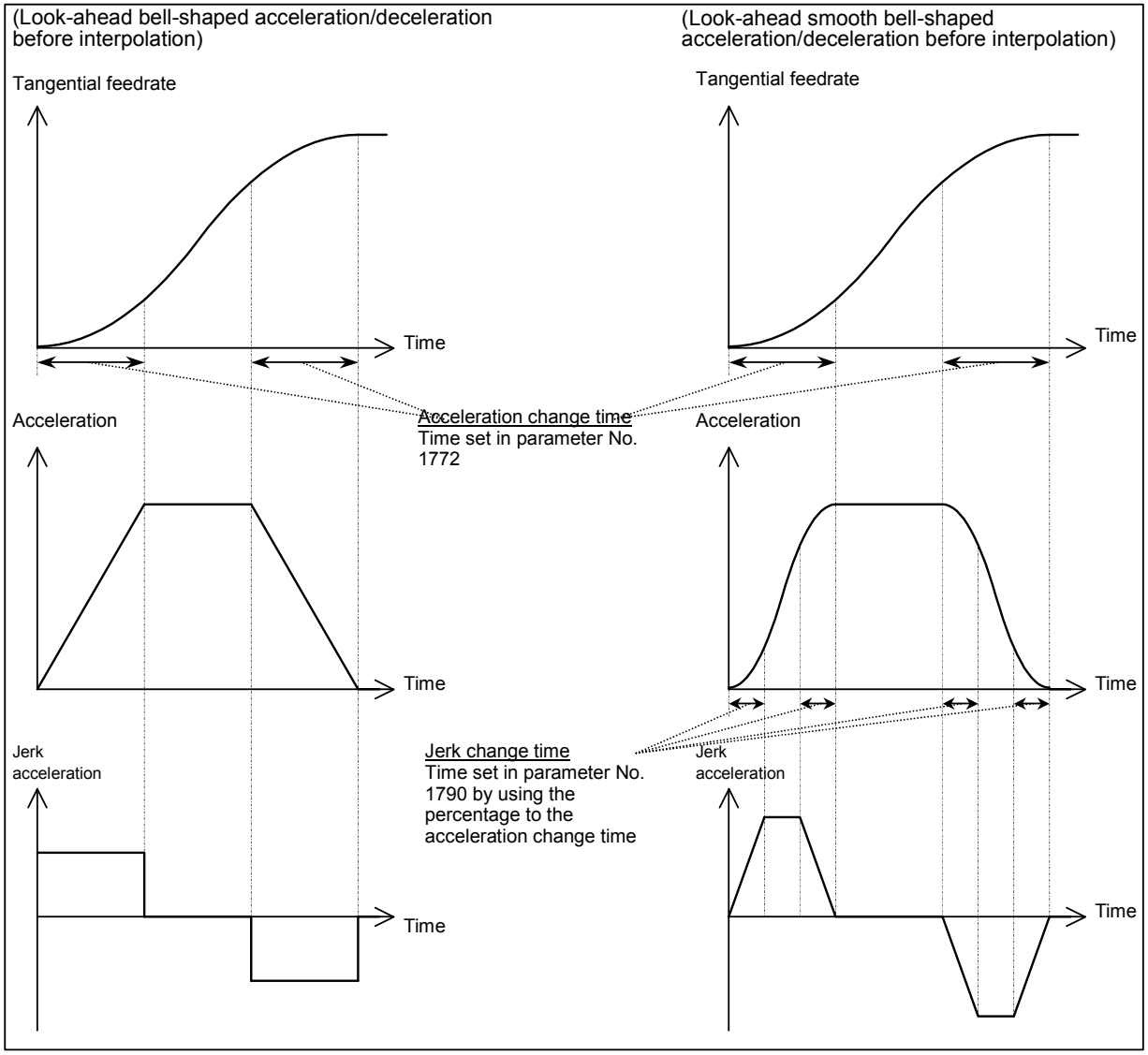
7.3.2 Look-Ahead Smooth Bell-Shaped Acceleration/Deceleration before Interpolation

Overview

In look-ahead bell-shaped acceleration/deceleration before interpolation performs smooth acceleration/deceleration by changing the acceleration at a constant rate in specified acceleration change time.

In look-ahead smooth bell-shaped acceleration/deceleration before interpolation, the jerk change time is specified in parameter No. 1790 by using the percentage to the acceleration change time for look-ahead bell-shaped acceleration/deceleration before interpolation, and change of acceleration is also controlled so that the change is bell-shaped.

This enables smoother acceleration/deceleration, therefore it reduces machine vibration and shock due to acceleration/deceleration.



Explanation

- Setting the jerk change time

The jerk change time is set in parameter No. 1790 by using the percentage to the acceleration change time.
The actual jerk change time is represented by the percentage to the acceleration change time set in parameter No. 1772.
The jerk change time must be within a half of the acceleration change time, so the value to be set in the parameter ranges 0 to 50 (percent).
If 0 or a value beyond the specifiable range is specified in parameter No. 1790, look-ahead smooth bell-shaped acceleration/deceleration before interpolation is not available.

- Acceleration/deceleration before interpolation for linear type rapid traverse

When bell-shaped acceleration/deceleration is used in acceleration/deceleration before interpolation for linear type rapid traverse, enabling look-ahead smooth bell-shaped acceleration/deceleration before interpolation applies smooth bell-shaped acceleration/deceleration to acceleration/deceleration before interpolation for linear type rapid traverse.
In this case, the jerk change time is represented by the percentage set in parameter No. 1790 to the acceleration change time set in parameter No. 1672.

Parameter

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.

Note

NOTE
To use feedrate control based on an acceleration change of each axis, the acceleration control option and AI contour control II option are required.

8 2-PATH CONTROL

T

Chapter 8, “2-PATH CONTROL”, consists of the following sections:
This chapter describes 2-path control for the T series.

8.1 2-PATH CONTROL 578
8.2 WAITING M CODES.....591
8.3 PATH INTERFERENCE CHECK 593
8.4 BALANCE CUTTING 602
8.5 SYNCHRONOUS CONTROL AND COMPOSITE CONTROL 605
8.6 SUPERIMPOSED CONTROL 650
8.7 SYNCHRONOUS, COMPOSITE, AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND 663
8.8 PATH SPINDLE CONTROL 665
8.9 MEMORY COMMON TO PATHS 675
8.10 PATH SINGLE BLOCK CHECK FUNCTION 678
8.11 PATH SELECTION/DISPLAY OF OPTIONAL PATH NAMES 679

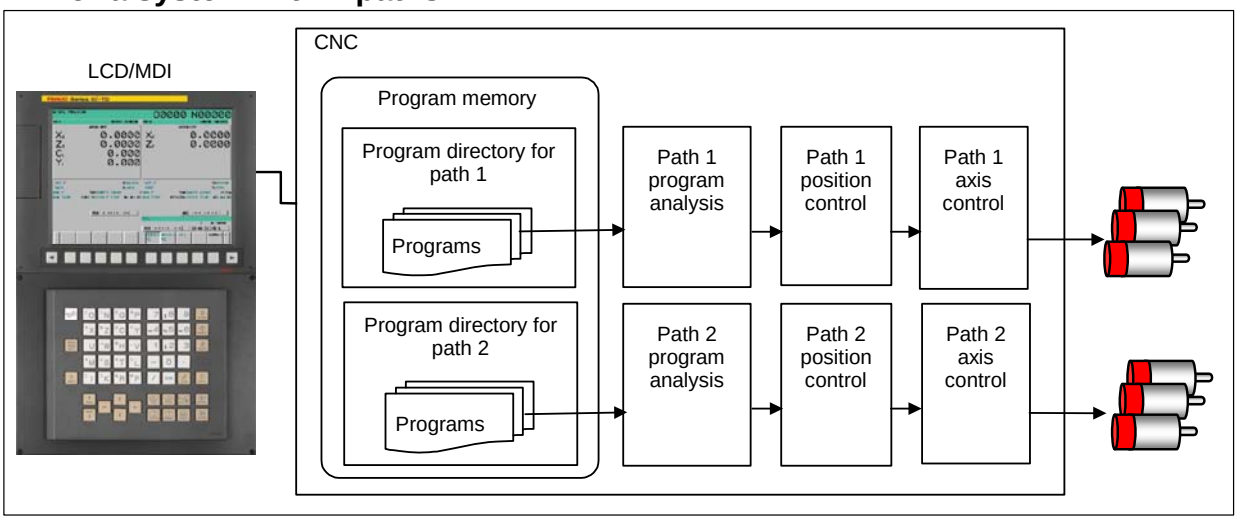
8.1 2-PATH CONTROL

Overview

The 2-path control function is designed to enable 2 independent simultaneous machining with up to 2 paths.
For 2-path simultaneous machining, each machining program is stored in a directory 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 path 2 from the programs stored in the respective directories 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 2-path control include inter-path interference check, balance cut, synchronization control, composite control, path spindle control, and memory common to paths.
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.

NOTE
2-path control function (T series) is optional function.

- For a system with 2 paths

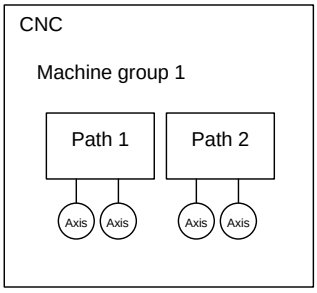


Explanation

- System configuration

In 2-path control, two paths are combined to comprise a single system.
Each of the elements making up 2-path control is explained below.

Example)
Configuration of 2-path control



- Path

A path refers to a group of axes controlled by the same CNC program command.
You can have up to 2 paths available.

- Controlled axes

You can have up to 9 controlled axes available throughout the system. (Maximum 7 axes per one-path)

NOTE
1 The maximum numbers above vary with the option configuration. For details, refer to the manual provided by the machine tool builder.
2 The number of the spindles may limit the maximum number of the controlled axes(=feed axes). (Limitation by the total number of control axes)
See 1.1 “Controlled axis” for details.

- Setting of the number of axes

Set the number of axes to be controlled for each path by using parameter No. 8130. If the total number of controlled axes exceeds the maximum value in a multi-path system, the number of controlled axes is clamped at the maximum number of axes set for the system.

When spindle control with servo motor is enabled, the maximum number of controlled axes is increased by one for both the number of axes for each path and the total number of axes. For the path containing the tool rotation axis, set the number of axes including the tool rotation axis. For information about spindle control with servo motor, see "SPINDLE CONTROL WITH SERVO MOTOR".

- Assignment of axes to paths

You can specify which path each controlled axis is to be assigned, using parameter No. 981.
Controlled axes may be assigned to each path arbitrarily as long as the number of axes is within the allowable ranges of the total number of controlled axes and the number of controlled axes (on a per-path basis).

Example 1)

For a system with 2 paths and a total of 6 controlled axes:
Settings that cause the system to become a 2-path system with the first path consisting of 3 controlled axes and the second consisting of 3 controlled axes

Parameter No. 0981 setting	Application
1	Path 1, axis 1
1	Path 1, axis 2
1	Path 1, axis 3
2	Path 2, axis 1
2	Path 2, axis 2
2	Path 2, axis 3

Example 2)

For a system with 2 paths and a total of 7 controlled axes:
Settings that cause the system to become a 5-path system with the first path consisting of 2 controlled axes and the second consisting of 3 controlled axes

Parameter No. 0981 setting	Application
1	Path 1, axis 1
1	Path 1, axis 2
1	Path 1, axis 3
1	Path 1, axis 4
1	Path 1, axis 5
2	Path 2, axis 1
2	Path 2, axis 2

NOTE
1 If the setting of parameter No. 981 is 0, that controlled axis will belong to the first path.
2 If the assignment of axes to paths specified by parameter No. 0981 does not match the number of controlled axes set for each path by parameter No. 8130, an alarm PS0365 occurs.
3 If this alarm PS0365 occurs, the system comes to an emergency stop. In this case, set an appropriate value in parameter No. 0981 and then turn the power back on.

- Automatic setting

If all the values set in parameter No. 0981 are 0, the parameter is automatically set based on the number of controlled axes set for each path.

- Spindle

In a 2-path control system, the maximum number of spindles is 4. (Maximum 3axes per one-path)

NOTE
1 The maximum numbers above vary with the option configuration. For details, refer to the manual provided by the machine tool builder.
2 The number of the controlled axes(=feed axes) may limit the maximum number of the spindles. (Limitation by the total number of control axes)
See 1.1 “Controlled axis” for details.

- Setting of the number of spindles

- When serial spindle output is enabled (0 is set in bit 5 (SSN) of parameter No. 8133)
  Based on bit 1 (ISI) and bit 4 (SS2) and bit 5 (SS3) of path-related parameter No. 3701, the number of spindles in each path is set as follows.

SS3	SS2	ISI	Number of spindles in the path
"0"	"0"	"1"	0
"0"	"0"	"0"	1
"0"	"1"	"0"	2
"1"	"0"	"0"	3

- When serial spindle output is disabled (1 is set in bit 5 (SSN) of parameter No. 8133)
  One spindle is assigned to each path, and one analog spindle can be controlled. In this case, the spindle of the second path is not used; therefore, set 0 in parameter No. 3717 when setting the amplifier number of the spindle.
  Note that bit 1 (ISI) and bit 4 (SS2) and bit 5 (SS3) of parameter No. 3701 become invalid.

When spindle control with servo motor is enabled, the servo motor used as the tool rotation axis is regarded as one of the spindles. Therefore, when setting the number of spindles, include the tool rotation axis. The number of spindles that can be connected is decreased by the number of tool rotation axes.
If the total number of spindles exceeds the maximum value, an alarm PS1996 occurs.

- Assignment of spindles to paths

You can specify which path each spindle is to be assigned, using parameter No. 982.
You can assign each path to any controlled axis, provided that you follow the maximum allowable number of spindles.
Example 1)
In the case of a 2-path control system with one spindle assigned to both path 1 and path 2

Parameter No. 0982 setting	Application
1	Path 1, spindle 1
2	Path 2, spindle 1

Example 2)
In the case of a 2-path control system with two spindles assigned to path 1 and no spindles assigned to path 2

Parameter No. 0982 setting	Application
1	Path 1, spindle 1
1	Path 1, spindle 2

Example 3)
In the case of a 2-path control system with two spindles assigned to path 1 and one spindle assigned to path 2

Parameter No. 0982 setting	Application
1	Path 1, spindle 1
1	Path 1, spindle 2
2	Path 2, spindle 1

Example 4)

In the case of a 2-path control system with one spindle assigned to path 1 and two spindles assigned to path 2

Parameter No. 0982 setting	Application
1	Path 1, spindle 1
2	Path 2, spindle 1
2	Path 2, spindle 2

NOTE

1 If the setting of parameter No. 982 is 0, that spindle will belong to the first path.
2 If the assignment of spindles to paths specified by parameter No. 0982 does not match the number of spindles set for each path by bit 1 (ISI) and bit 4 (SS2) and bit 5 (SS3) of parameter No. 3701, an alarm PS0365 occurs.
3 If this alarm PS0365 occurs, the system comes to an emergency stop. In this case, set an appropriate value in parameter No. 0982 and then turn the power back on.

- Automatic setting

If all the values set in parameter No. 0982 are 0, the parameter is automatically set based on the number of spindles set for each path.

- Signals common to paths

These signals are common to paths.

Path select

Path select signal	HEAD<G063.0>

Auto screen clear function

Auto screen clear invalidation signal	*CRTOF<G062.1>

Screen hard copy function

Hard copy cancellation request signal	HCABT<G067.6>
Hard copy cancellation request acceptance signal	HCAB2<F061.2>
Hard copy execution request signal	HCREQ<G067.7>
Hard copy under execution signal	HCEXE<F061.3>

External key input function

External key input mode select signal	ENBKY<G066.1>
Key code read signal	EKSET<G066.7>
Key code signal	EKC0 to EKC7<G098.0to G098.7>
Program screen display in progress signal	PRGDPL<F053.1>
Key code read completion signal	EKENB<F053.7>
Key input nullification signal	INHKY<F053.0>

NOTE

The signal area is common to paths. Key input is made on the currently selected screen.

Memory protection function

Memory protection signal KEYP, KEY1 to KEY4 <G046.0, G046.3 to G046.6>
Memory protect signal KEYO<F075.6>

8.1.1 CNC Data Display, Setup, and Input/Output

There are offsets and custom macro variables for each path. For the path selected from the display unit, you can perform data display, setup, and input/output using a medium such as a memory card.
As for custom macro variables, you can change some or all of common variables into those for common use by paths, through appropriate parameter settings.
As for offsets, you can change them into those for common use by paths, through appropriate parameter settings.
As for input/output, you can rename the files to be input and output for each path, through appropriate parameter settings.

8.1.2 2-path Functions

The functions specific to 2-path control are as listed below.

Waiting M code
Path interference check
Balance cutting
Axis recomposition
Superimposed control
Path spindle control
Memory common to paths
Path single block check function
Path selection/arbitrary path name display

The following provides an outline of each of the functions.

- Waiting M code

M codes control the first and second paths so that one waits for the other during machining.
By specifying a waiting M code in the machining program for each of two paths, you can synchronize these paths. If, during automatic operation, a waiting M code is specified for one path, the program waits for the same M code to be specified for another path, then starts executing the next block.
You must specify the range of M codes that can be used as waiting M codes in advance, using an appropriate parameter.
For details, see "WAITING M CODES".

- Path interference check

If you machine a single workpiece on two tool posts at the same time, the portions of these tool posts may come close to each other exceedingly. If these two tool posts come into contact due to a program error or other setting error, this will cause grave consequences such as the damage of the tool or the damage of the machine itself.
The "check for tool post interference" refers to the function whereby if a command that will cause two tool posts to interfere with each other as described above is specified, these two tool posts are both decelerated to a stop before they actually come into contact.
For details, see "Path Interference Check".

- Balance cutting

If turning a thin workpiece, you can machine it from both of its sides at the same time by using a cutting tool, thereby preventing the workpiece from deflection, which can occur if it is machined from a single side only, and achieving higher precision. Unless the movements of the two tools are synchronized with each other well, the workpiece will vibrate, making correct machining impossible. By using this function, you can easily synchronize the movements of the tool posts and execute machining.
For details, see "BALANCE CUTTING".

- Synchronous/Composite control

Usually, in 2-path control, the tool moves along the axes (X1, Z1, and C1) belonging to tool post 1 in accordance with the move commands for tool post 1 and along the axes (X2, Z2, and C2) belonging to tool post 2 in accordance with the move commands for tool post 2 (independent control for each tool post). The synchronous/composite control function allows you to move the tool along any of the axes of one tool post in synchronization with any of the axes of the other tool post (synchronous control). The function also allows you to replace the move command for any of the axes of one tool post and that for any of the axes of the other tool post with other to move the tool along these axes with these commands (composite control).
For details, see "SYNCHRONOUS/COMPOSITE CONTROL".

- Superimposed control

Usually, in 2-path control, the tool moves along the axes (X1, Z1, and C1) belonging to tool post 1 in accordance with the move commands for tool post 1 and along the axes (X2, Z2, and C2) belonging to tool post 2 in accordance with the move commands for tool post 2 (independent control for each tool post). The superimposed control function allows you to move the tool along any of the axes of one tool post and along any of the axes of the other tool post by superimposing the amount of travel.
For details, see "SUPERIMPOSED CONTROL".

- Path spindle control

This function allows the spindle belonging to one path to follow the spindle command of another specific path.
It allows you to specify that a path is to incorporate the position coder feedback data for any of the spindles belonging to any path. This achieves control using a spindle of another path, for example, threading or feed per revolution in path 1, using a spindle belonging to path 2.
For details, see "PATH SPINDLE CONTROL".

- Memory common to paths

In a machine with multiple paths, this function allows you to make common the independent custom macro common variables and tool compensation memory of a path, through appropriate parameter settings.

- Custom macro common variables

Part or all of common variables #100 to #149 and #500 and #531 of custom macros can be made into those for common use by paths (variables that permit reading or writing by either of both paths).

- Tool compensation memory

The tool compensation memory of path 1 can be accessed by path 2 as well, for data reference and setting.
The requirement is that the settings for tool compensation (such as the number of sets, the number of digits, and the unit system) must be the same in both paths 1 and 2.
For details, see "MEMORY COMMON TO PATHS".

- Path single block check function

If one path enters the single block stop state, this function allows another path to enter the feedhold stop state, enabling single block operation with the path machining programs being nearly synchronized.
For details, see the "PATH SINGLE BLOCK CHECK FUNCTION".

- Path selection/arbitrary path name display

Path selection/arbitrary path name display
This function is used to select a path targeted for an operation that is performed for a specified path, for example, displaying and setting various types of data (tool compensation or the like) for each path, inputting the command program in MDI mode, or editing the machining program in program memory.

In addition, the name of each path can be changed by setting the parameter.
For details, see "Path selection/arbitrary path name display".

8.1.3 Cautions on 2-path Control

- Parameter

> ⚠ **WARNING**
> If you attempt to change parameters in MDI mode in a certain path, it is probable that other paths are in MEM or other modes even if that path is in MDI mode. Before changing parameters, check to see if this will not affect the operations in the other paths.
> During axial movement, in particular, use great caution not to change those parameters related to axes and spindles.

- MDI reset key

> ⚠ **CAUTION**
> Only a single LCD/MDI set is provided for all paths. You use the path selection signal to specify which path the operations and displays on the LCD/MDI are for. The reset key on the MDI panel is, however, effective to all paths, regardless of the path selection signal.
> Note, however, that you can make the key effective to the path selected with the path selection signal, by appropriately setting parameter No.8100#0 (RST).

- Alarms during automatic operation

> ⚠ **CAUTION**
> If, during automatic operation, an alarm occurs in a certain path, all paths in that machine group will enter the feedhold state and stop.
> You can let them continue to operate, disregarding the alarm, by appropriately setting parameter No.8100#1(IAL).

Parameter

0981	Absolute path number to which each axis belongs

> **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] 1, 2
Set the path to which each axis belongs.

> **NOTE**
> 1 When 0 is set for all axes, the parameter is automatically set according to the number of controlled axes of each path.
> 2 When the setting falls outside the range, the axis is assumed to belong to the first path.

0982	Absolute path number to which each spindle belongs

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] 1, 2
Set the path to which each spindle belongs.

NOTE
1 When 0 is set for all axes, the parameter is automatically set according to the number of controlled axes of each path.
2 When the setting falls outside the range, the axis is assumed to belong to the first path.
3 When spindle control with servo motor is enabled, the servo motor used as the spindle controlled axis is treated as a spindle. Therefore, it is necessary to set the path to which the axis subject to spindle control with servo motor.

	#7	#6	#5	#4	#3	#2	#1	#0
3008						XSG		

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#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 327
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 option configuration of the I/O Link, the actually usable X addresses are:
X0 to X127, X200 to X327

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 327
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 option configuration of the I/O Link, the actually usable X addresses are:
X0 to X127, X200 to X327

3014 **Bit position of an 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] Byte axis
[Valid data range] 0 to 7
Set a bit position to which the deceleration signal for reference position return (*DECn) for each axis is to be assigned.

NOTE
This parameter is valid when bit 2 (XSG) of parameter No. 3008 is set to 1.

3019	Address to which the PMC axis control skip signal and measurement position arrival 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 327

Set an X address to which the PMC axis control skip signal ESKIP and the measurement position arrival signals (XAE1, XAE2, and XAE3 (M series) or XAE1 and XAE2 (T series) are to be assigned.

Example 1. When No.3012 is set to 5 and No.3019 is set to 6

When XSG (bit 2 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	
X005	SKIP	SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	(T series)
	SKIP	SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	(M series)

	#7	#6	#5	#4	#3	#2	#1	#0	
X006		ESKIP					XAE2	XAE1	(T series)
		ESKIP				XAE3	XAE2	XAE1	(M series)

Example 2. When No.3012 is set to 5 and No.3019 is set to 5

When XSG (bit 2 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	
X005	SKIP	ESKIP					XAE2	XAE1	(T series)
		SKIP6	SKIP5	SKIP4	SKIP3	SKIP2	SKIP8	SKIP7	
	#7	#6	#5	#4	#3	#2	#1	#0	
	SKIP	ESKIP	SKIP5	SKIP4	SKIP3	XAE3	XAE2	XAE1	(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 option configuration of the I/O Link, the actually usable X addresses are:
X0 to X127, X200 to X327

	#7	#6	#5	#4	#3	#2	#1	#0
3701			SS3	SS2			ISI	

[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.

#1 ISI
#4 SS2
#5 SS3 Set the number of spindles in a path.

SS3	SS2	ISI	Number of spindles in a path
0	0	1	0
0	0	0	1
0	1	0	2
1	0	0	3

NOTE
This parameter is valid when spindle serial output is enabled (bit 5 (SSN) of parameter No. 8133 is 0).

	#7	#6	#5	#4	#3	#2	#1	#0
8100		DSB					IAL	RST

[Input type] Parameter input
[Data type] Bit machine group

#0 RST The pressing of the RESET key on the MDI panel is:
0: Valid for two paths.
1: Valid only for the path selected by the path selection signal.

#1 IAL Choice of an option concerning operation continuation when an alarm is issued, and choice of an option concerning the start of automatic operation in alarm state:
0: - When an alarm is issued, the operation is stopped with the other path(s) placed in hold state.
- When the other path or paths are placed in alarm state, automatic operation cannot be started.
1: - Even when an alarm is issued, the operation is continued without stopping the other path(s).
- Even when the other path or paths are placed in alarm state, automatic operation can be started.

#6 DSB The inter-path single block check function is:
0: Disabled.
If one of the paths undergoes a single block stop, only the path undergoes a single block stop.
1: Enabled.
If one of the paths undergoes a single block stop, the other path undergoes a feed hold stop.

8130	Number of controlled 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] 1 to Maximum number of controlled axes
This parameter sets the number of axes for each path.

NOTE
When spindle control with servo motor is enabled, set the number of axes including this axis for the axes with a spindle controlled axis with servo motor.

Alarm and Message

Number	Message	Description
PS0365	TOO MANY MAXIMUM SV/SP AXIS NUMBER PER PATH	The Max. total number of control axes, the Max. number of feed axes or the Max. number of control axes is exceeded. Check parameter No. 0981 and No. 0982. If this alarm occurs, the emergency stop cannot be canceled.
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.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (This manual)	Controlled axis
CONNECTION MANUAL (FUNCTION) (This manual)	Designation of spindle axes

8.2 WAITING M CODES

Overview

M codes control the first and second paths so that one waits for the other during machining. When a waiting M code 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.
Also, by using bit 0 (MWT) of parameter No. 8103, it is possible to select the path common signal interface (MWT = 1) or path individual signal interface (MWT = 0).

Signal

You can invalidate waiting by using the waiting invalidation signal.
Use this signal to perform automatic operation on one path while holding the other path when running an automatic operation program that contains waiting M codes.
When the path individual signal interface is used (0 is set in bit 0 (MWT) of parameter No. 8103), it is necessary in principle to set "1" in the no-wait signal for the path to be held.

No-wait signal NOWT<G0063.1>(for path common signal interface) NMWT<Gn063.7>(for path individual signal interface)

[Classification] Input signal
[Function] Specifies whether to synchronize the paths by the waiting M code.
[Operation] When this signal turns to "1" the paths are not synchronized by the M code. The M code for waiting specified in a machining program is ignored.
When this signal turns to "0", the paths are synchronized by the M code. When the waiting M code is specified for one path, the CNC waits for the corresponding M code of another path to be issued, then starts executing the next block.

Waiting signal WATO<F0063.6>(for path common signal interface) WATO<Fn063.6>(for path individual signal interface)

[Classification] Output signal
[Function] Indicates that the M code is waiting for a path.
[Output cond.] These signals are "1" as long as:
- One path is waiting for another path. That is, the signal stays "1" for the period from when the M code for waiting is issued to one path to when the corresponding M code is issued to another path.

This signal is "0" as long as:
- Neither of the paths are waiting for the other.

NOTE
When the path common signal interface (bit 0 (MWT) of parameter No. 8103 = 1) is used, F0063.6="1" results even if path 2 is placed in the wait state.
So, when the wait state needs to be checked with each path, use the path individual signal interface (bit 0 (MWT) of parameter No. 8103 = 0).

Signal address

- For path common signal interface (bit 0 (MWT) of parameter No.8103 =1)

	#7	#6	#5	#4	#3	#2	#1	#0
G0063							NOWT	
	#7	#6	#5	#4	#3	#2	#1	#0
F0063		WATO						

- For path individual signal interface (bit 0 (MWT) of parameter No.8103 =0)

	#7	#6	#5	#4	#3	#2	#1	#0
Gn063	NMWT							
	#7	#6	#5	#4	#3	#2	#1	#0
Fn063		WATO						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8103								MWT

[Input type] Parameter input
[Data type] Bit

> **NOTE**
> When this parameter 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 signal interface for three paths is used.
1: The conventional signal interface for two paths is used.

> **NOTE**
> When the path common signal interface is used (1 is set in bit 0 (MWT) of parameter No. 8103), the waiting signal (WATO<F0063.6>) is set to "1" even if path 2 is placed in the wait state.
> Therefore, when the wait state needs to be checked with each path, use the path individual signal interface (bit 0 (MWT) of parameter No. 8103 = 0).

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 coder 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.

Alarm and message

Number	Message	Description
PS0160	MISMATCH WAITING M-CODE	Different waiting M codes are specified for path 1 and path 2. Modify the program.

Alarms occur in both paths, causing a single block stop and bringing the automatic operation to a halt.

Caution

⚠ CAUTION

1 Be sure to specify a waiting M code in an independent block.
2 Unlike other M codes, waiting M codes are not output to PMCs.
3 If you want to independently operate one path only, you need not delete that waiting M code. Using the waiting invalidation signal, you can invalidate the waiting M code specified in the machining program. For details, refer to the manual issued by the machine tool builder.
4 To use a waiting M code in mode for multiple M commands in a single block, the M code must be specified as the first M code.

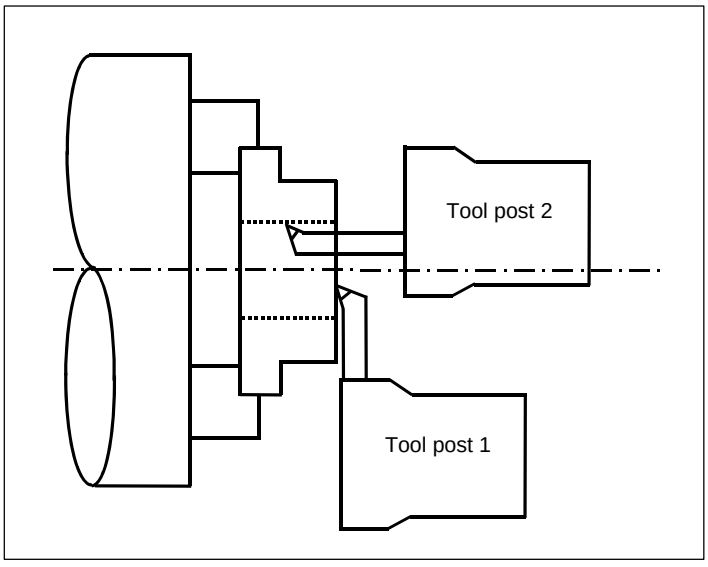
Reference item

Manual name	Item name
OPERATOR'S MANUAL (Lathe system) (B-64304EN-1)	Waiting for paths

8.3 PATH INTERFERENCE CHECK

Overview

When tool posts on individual paths machine the same workpiece simultaneously, the tool posts can approach each other very closely. If the tool posts interfere with each other due to a program error or any other setting error, a serious damage such as a tool or machine destruction can occur.
The function "path interference check" decelerates and stops the tool posts on individual paths before they actually make contact with each other if a command is issued and they will interfere with each other as the result.



The contours and shapes of the tool posts on individual paths are checked to determine whether or not an interference occurs.

NOTE
Path interference check is optional function.

Explanation

- Data Setting for the Path Interference Check Function

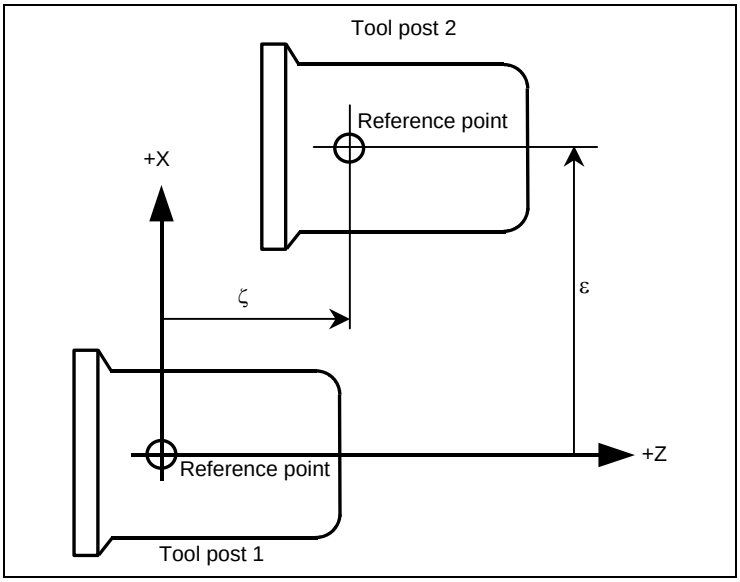
To make a path interference check, data including the relationships between the tool posts on individual paths and interference forbidden areas (that is, tool shapes) needs to be set. The method of such data setting is described below.

- Setting the positions for reference points of tool posts on individual paths

X described below is the axis for which 1 (X-axis of the three basic axes) is set in parameter No. 1022 and Z is the axis for which 3 (Z-axis of the three basic axes) is set in the parameter. If an invalid value is set in this parameter, a path interference check is not made.

- Setting the reference points for a path interference check

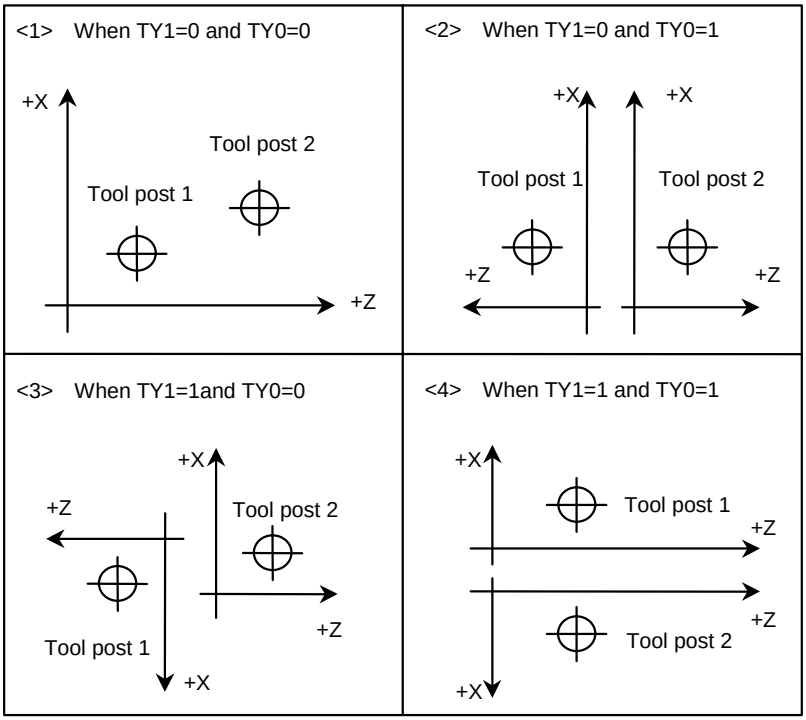
In the parameter, set the reference point of tool post 2 in the ZX plane coordinate system whose origin is the reference point of tool post 1 with all the axes (X1, Z1, X2, and Z2) returned to their respective reference points.



In the ZX plane coordinate system at the origin of which the reference point of tool post 1 is set, set the X coordinate (ε) of the reference point of tool post 2 in parameter No. 8151, and its Z coordinate (ζ) in parameter No.8152.
The unit of setting is the least command increment. For an axis subject to diameter specification, a diameter value is to be specified.
Measure (ε) and (ζ) when reference point return operation is completed with the four axes (X1, Z1, X2, Z2) (when the tools are in their reference points). When relative coordinate parameters Nos. 8151 and 8152 of the tool posts are to be updated, reference point return operation must always be completed with the four axes beforehand. Otherwise, the internally memorized relational positions of the tool posts are not updated to new parameter values.

- **Setting the relationship between the coordinate systems for a path interference check**

Set the relationship between the coordinate systems of the two tool posts using TY0 and TY1 (bits 0 and 1 of parameter No.8140), with tool post 1 used as the reference.



Display and setting of an interference forbidden area in a 2-path interference check

Display and set tool figure data (interference forbidden area) by performing the following operations:

<1> With the path selection signal, select a path for which an interference forbidden area is to be displayed and set in a 2-path interference check.

<2> Press the function key .

<3> Press the chapter selection soft key [TOOL FIGURE].

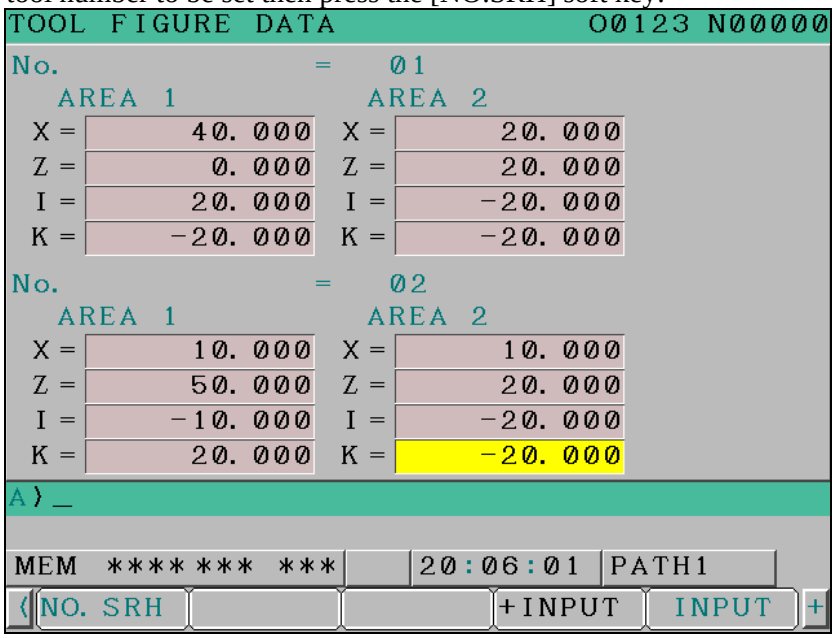
<4> Display the screen including a tool number for which data is to be set.

Method 1:

Switch the screen display by pressing the page key or cursor key.

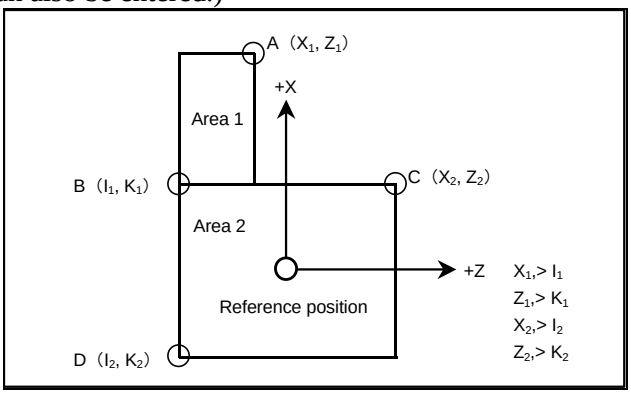
Method 2:

Enter a tool number to be set then press the [NO.SRH] soft key.



<5> Using the cursor key, place the cursor on the data to be set.
(When setting the data of point A, move the cursor to X and Z in AREA 1. When setting the data of point B, move the cursor to I and K in AREA 1.)

<6> Using the numeric keys, enter the coordinates of points A, B, C, and D.
(The decimal point can also be entered.)



<7> By pressing the [INPUT] soft key, the entered coordinate values are set.
(To add an entered value to the already set data, press the [+INPUT] soft key.)

NOTE

1 Tool number
Tool figure data needs to be set for each tool number. A tool number used here means an offset number. When both of geometry offset and wear offset are used, a tool number corresponds to a wear offset number. When multiple offset numbers are used for the same tool, the same data needs to be set as tool figure data.

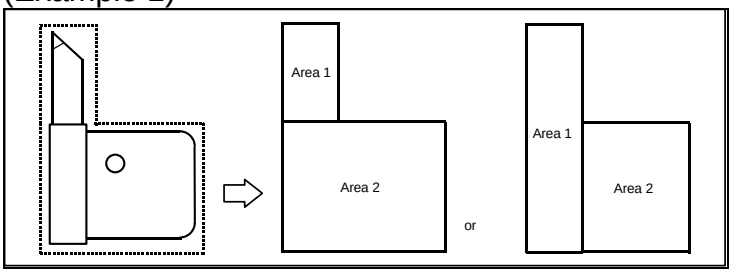
NOTE

2 Setting the contact prohibited area

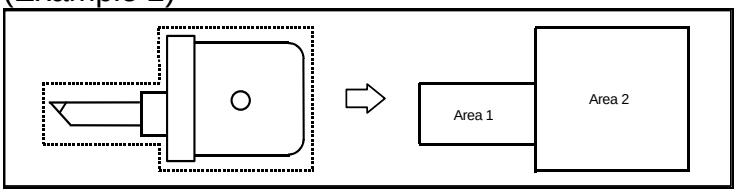
The contact prohibited area can be set using two rectangular areas in combination.

Examples follow.

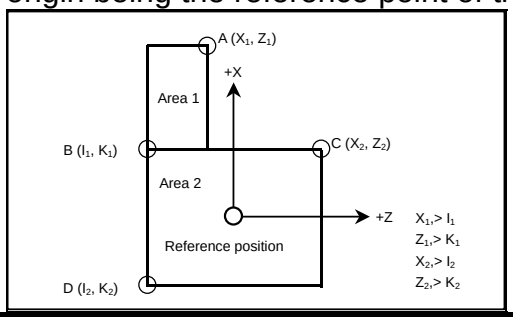
(Example 1)



(Example 2)



With regard to each of the two rectangles, set the coordinates of both the upper end and lower end of the rectangle (A, B, C, and D in the figure below), with the origin being the reference point of the tool post.



- Conditions for Making a Path Interference Check

A path interference check is made when conditions listed below are satisfied.

(1) Common conditions

- The inter-path interference check is enabled (0 is set in bit 4 (IFE) of parameter No. 8140).
- The X- and Z-axes of the three basic axes are set for parameter No. 1022 for individual paths for which a path interference check is to be made.
- The basic axes of both paths have returned to their respective reference points after powering on (when the absolute position detector is used, the correspondence between machine positions and absolute position detector-detected positions has been established).
- Offset numbers other than 0 are specified using T codes for tool posts on the paths for which a path interference check is made.
- Values are input in either one or both of the areas of the tool shape data corresponding to the specified tool number and the relationships among the values are X > I and Z > K.
- When manual mode is used, parameter IFM (No. 8140#3) for enabling the interference check for each path in manual mode is set to 1.

(2) When the interference check conditions are satisfied

When all conditions for making a path interference check are satisfied, the path-interference-check-in-progress signal is output to the PMC.

> ⚠ **WARNING**
> To make a path interference check, the tool number for which tool shape data is input must agree with the programmed tool number. An interference check cannot be made correctly if the tool is selected by manual operation or if no tool select command is specified at power-on.

Signal

Path interference check in progress signal TICHK<F064.6>

[Classification] Output signal
[Function] Informs whether a path interference check is in progress.
[Output cond.] This signal goes to "1" when:

- All conditions necessary to perform a path interference check on the tool posts of the first and second paths have been satisfied.

This signal goes to "0" when:

- Any one of the conditions necessary to perform a path interference check on the tool posts of the first and second paths have not been satisfied.

Path interference alarm signal TIALM<F064.7>

[Classification] Output signal
[Function] Informs whether an alarm has occurred due to a path interference check.
[Output cond.] This signal goes to "1" when:

- The two tool posts are judged to interfere with each other while a path interference check is in progress on the tool posts of the first and second paths.

This signal goes to "0" when:

- The two tool posts are judged not to interfere with each other while a path interference check is in progress on the tool posts of the first and second paths.
- A path interference check is not performed (the path interference check signal TICHK is "0")

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F0064	TIALM	TICHK						

Caution

> **NOTE**
> 1 If the tool posts of the two paths are judged to have interfered during the inter-path interference check, both paths are decelerated and stopped and the control unit is placed in an alarm state. If this happens, the inter-path interference alarm signal TIALM<F064.7> is set to "1" to notify you that an inter-path interference check alarm has occurred.

NOTE

2 If an alarm occurs due to a path interference check, first switch to manual mode, manually retract the tool post of the path in which the alarm occurred to a position without interference, then perform a reset to cancel the alarm on the control unit. By manually retracting the tool post, the TIALM signal <F0064.7> will become "0" at the point when the tool post associated with the interference is judged not to interfere. (This can be used as a guideline: If, after a tool post has interfered, retracting it manually, you have only to retract it until the TIALM signal <F064.7> becomes "0".)

3 If an alarm occurs due to a path interference check, the axis and direction of movement that were assumed at that time will be stored, so that the tool can no longer be moved in the stored direction along the stored axis until the alarm on the control unit is canceled with a reset. (Any start that will result in further interference will be prohibited.)

4 If an interference alarm occurs, the tool post will stop before the interference position if parameter BFA (No.1300#7) is set to 1.
If this parameter is set to 0, the CNC and the machine system will stop somewhat later than the correct time, the positions at which they actually stop will be closer to the other tool post that the interference position specified in tool figure data. For safety, therefore, the tool figure data must be somewhat larger than necessary. The excess distance L may be calculated from the rapid traverse rate.
L = (Rapid traverse rate) x 1/7500
If, for example, the rapid traverse rate is 15 m/min, the figure data must be larger by about 2 mm.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA							

[Input type] Setting input
[Data type] Bit path

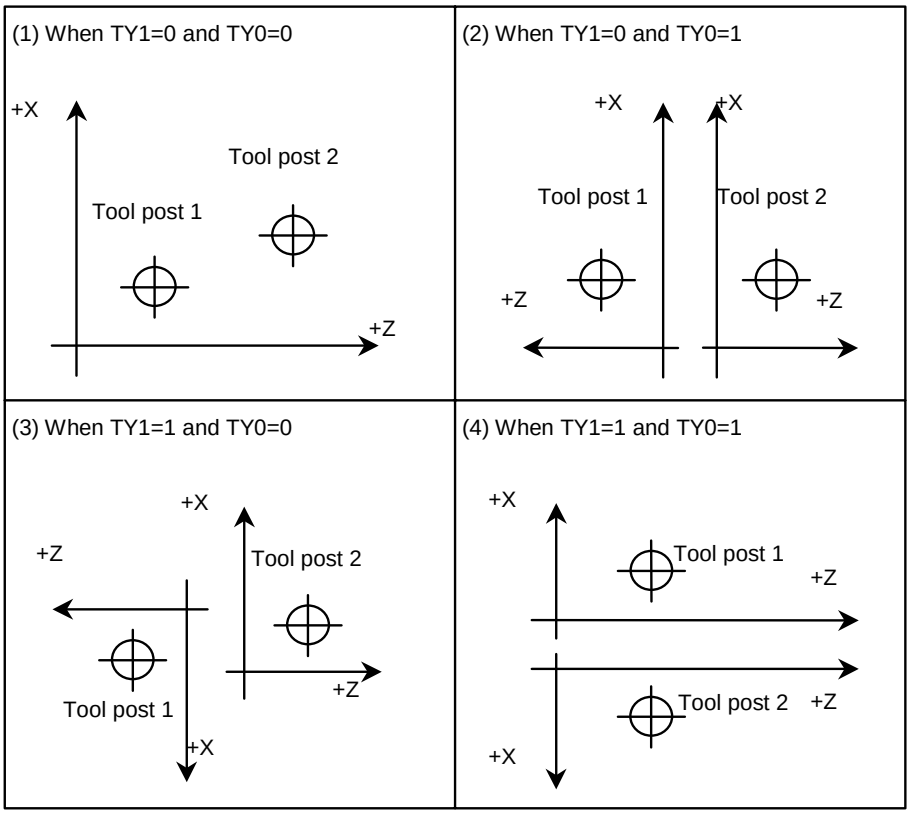
#7 BFA When the stored stroke check 1, 2, or 3 alarm is issued, an interference alarm is issued with the inter-path interference check function (T series), or a chuck/tail stock barrier (T series) alarm is issued:
0: The tool stops after entering the prohibited area.
1: The tool stops before the prohibited area.

	#7	#6	#5	#4	#3	#2	#1	#0
8140			ZCL	IFE	IFM	IT0	TY1	TY0

[Input type] Parameter input
[Data type] Bit

#0 TY0 This parameter sets the coordinate system relationship between two tool posts based on the tool post of path 1.

#1 TY1



#2 IT0 When offset number 0 is specified by the T code,

0: Checking interference between paths is stopped until an offset number other than 0 is specified by the next T code.

1: Checking interference between paths is continued according to the previously specified offset number.

#3 IFM In manual mode, a interference check between paths is:

0: Not performed.

1: Performed.

#4 IFE Interference check between paths is:

0: Performed.

1: Not performed.

#5 ZCL Specifies whether interference along the Z axis is checked while checking interference between paths.

0: Checked.

1: Not checked (Only interference along the X axis is checked.)

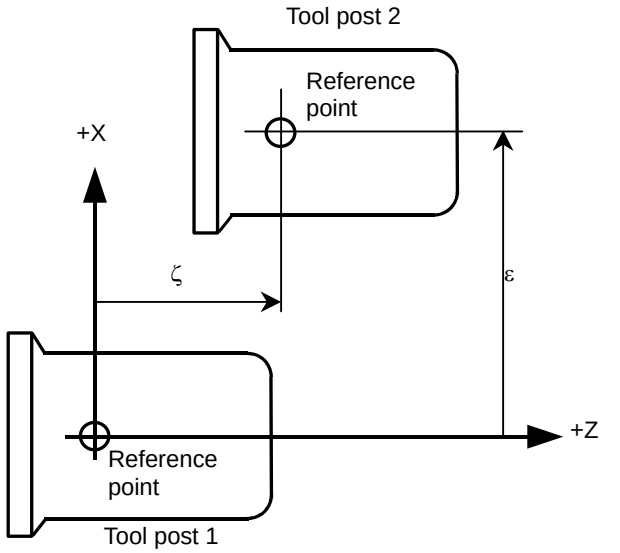
8151	Distance along the X axis between the reference positions of tool posts 1 and 2

8152	Distance along the Z axis between the reference positions of tool posts 1 and 2

[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 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)
Each of these parameters sets the distance between the tool posts of two paths.



In the Z-X plane coordinate system with its origin at the represent position of tool post 1, set the X component value ε of the reference position of tool post 2 in parameter No. 8151 and set the Z component value ζ in parameter No. 8152.

> ⚠ **WARNING**
> After modifying the parameter values, perform a manual reference position return operation for both tool posts. Otherwise, the internally stored positional relationships of the two tool posts are not updated to the newly set parameter values.

Alarm and message

Number	Message	Description
PS0169	ILLEGAL TOOL GEOMETRY DATA	Incorrect tool figure data in interference check. Set correct data, or select correct tool figure data.
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.

Alarms will occur individually for the axes associated with interference.

WARNING

⚠ WARNING

1 After setting parameters for the interference check function, tool figure data (contact prohibited area), etc., be sure to enter manual mode (parameter settings must be such that an interference check is enabled even in manual mode), let the tool post of each path interfere in all directions and confirm that a normal interference area has been set.

2 If an interference alarm occurs, the tool post will stop before the interference position when 1 is set in bit 7 (BFA) of parameter No. 1300.
When 0 is set in bit 7 (BFA) of parameter No. 1300, the CNC and the machine system will stop somewhat later than the correct time, resulting in the tool post actually stopping closer to the other tool post than the interference position specified in tool figure data. For safety reasons, therefore, the value set in the tool figure data must be somewhat larger than necessary. The excess distance L may be calculated from the rapid traverse rate.
L = (Rapid traverse rate) $\times$ 1 / 7500
For example, when the rapid traverse rate is 15 m/min, the value in the figure data must be larger by about 2 mm.

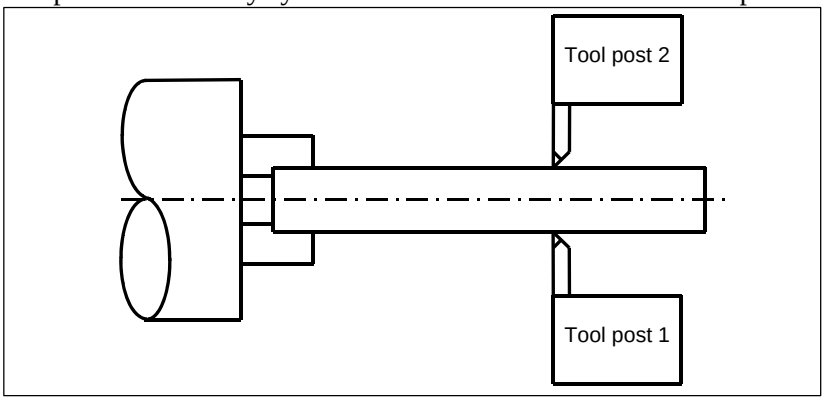
Reference item

Manual name	Item name
OPERATOR'S MANUAL (Lathe system) (B-64304EN-1)	Path interference check

8.4 BALANCE CUTTING

Overview

When a thin workpiece is to be machined as shown below, a precision machining can be achieved by machining each side of the workpiece with a tool simultaneously; this function can prevent the workpiece from warpage that can result when only one side is machined at a time (see the figure below). When both sides are machined at the same time, the movement of one tool must be in phase with that of the other tool. Otherwise, the workpiece can vibrate, resulting in poor machining. With this function, the movement of one tool post can be easily synchronized with that of the other tool post.



Specifying G68, which turns balance cutting mode on, causes balance cutting to be performed with the tool post of path 1 and that of path 2.

NOTE
1 Balance cutting is not performed in dry run or machine lock state. G68 or G69 specified for one tool post is synchronized with G68 or G69 specified for the other tool post, however.
2 In the balance cut mode, G68 specified for one tool post is not synchronized with G68 specified for the other tool post. In the balance cut cancel mode, G69 specified for one tool post is not synchronized with G69 specified for the other tool post.
3 Balance cutting is not performed in a block in which 0 is specified for the travel distance.
4 Balance cutting is not performed when rapid traverse is specified.
5 To use this function, set 0 in bit 0 (NVC) of parameter No. 8137. Also, when this function is used, mirror image for double turrets cannot be used. To use mirror image for double turrets, set 1 in bit 0 (NVC) of parameter No. 8137.
6 Balance cutting is optional function.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8137								NVC

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 NVC Balance cutting is:
0: Used.
1: Not Used.

NOTE
When balance cutting is used (this parameter is 0), the mirror image of facing tool posts cannot be used. To use the mirror image of facing tool posts, set this parameter to 1.

Caution

⚠ CAUTION
1 Balance cut only starts cutting feed on both tool posts at the same time; it does not maintain synchronization thereafter. To synchronize all the movements of both tool posts, the setting for both tool posts, such as the travel distance and feedrate, must be the same. Override and interlock can be applied independently to both tool posts. The settings for both tool posts that are related to override and interlock must also be the same to perform balance cutting.
2 After feed hold is applied during execution of balance cutting for both tool posts, balance cutting is not performed at the restart. Balance cutting is performed when the next move command is executed for both tool posts.

Note

NOTE

1 Time delay before the pulse distribution of both tool posts is started is 2 msec or shorter.
2 Overlap is invalid. In the balance cut mode, synchronization is established at the start of each move block in which cutting is specified, so movement can momentarily stop.
3 In the balance cut mode, continuous thread cutting overlap is also invalid. Perform continuous thread cutting in the balance cut cancel mode.
4 To establish synchronization at the start of pulse distribution in a block in which thread cutting is specified, the same Position coder must be selected.
5 The cancel mode (G69) is unconditionally set by a reset.
6 When the option "mirror image for double turrets" is selected, the balance cut function cannot be used. To use "mirror image for double turrets", set 1 in bit 0 (NVC) of parameter No. 8137 in order to disable "balance cut".

Alarm and message

Number	Message	Description
PS0163	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).

Reference item

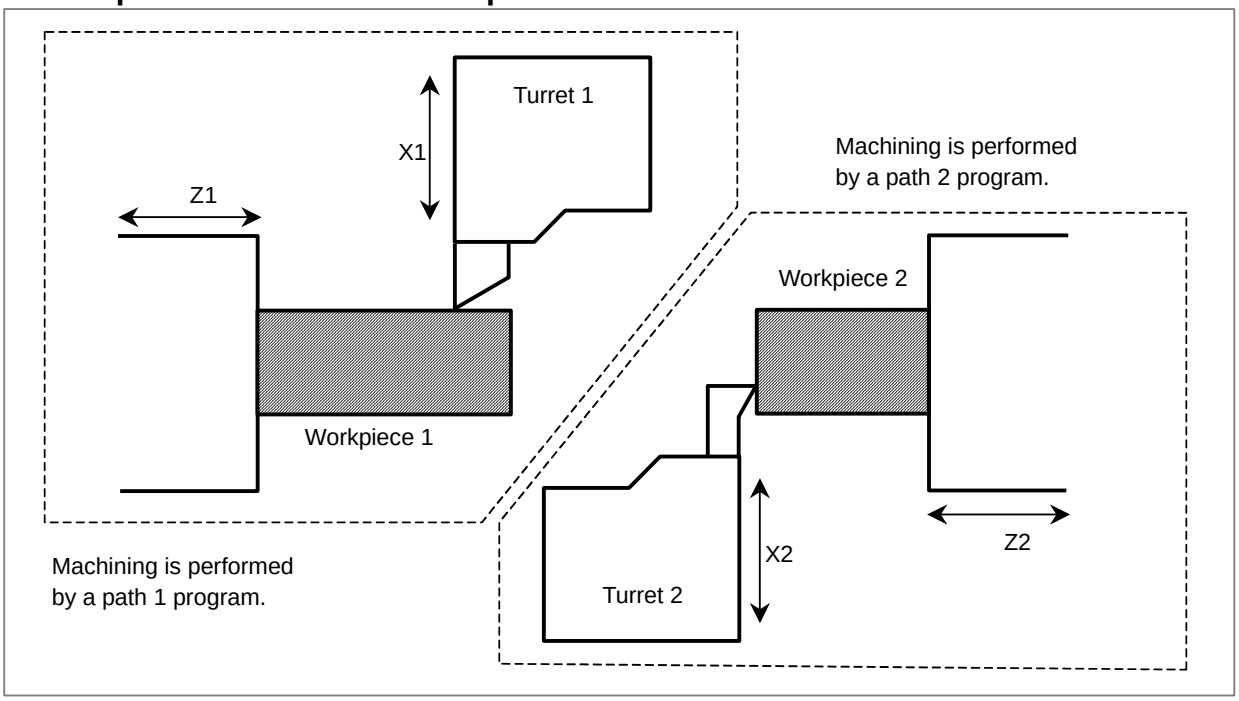
Manual name	Item name
OPERATOR'S MANUAL (Lathe system) (B-64304EN-1)	Balance cutting (G68,G69)

8.5 SYNCHRONOUS CONTROL AND COMPOSITE CONTROL

Overview

2-path control, which has multiple independent control paths built in, is used for such purposes as controlling multiple turrets of a complex lathe. The axes (such as X1-and Z1-axes) belonging to path 1 are controlled by commands in path 1, and the axes (such as X2- and Z2-axes) belonging to path 2 are controlled by commands in path 2.

- Independent control in each path

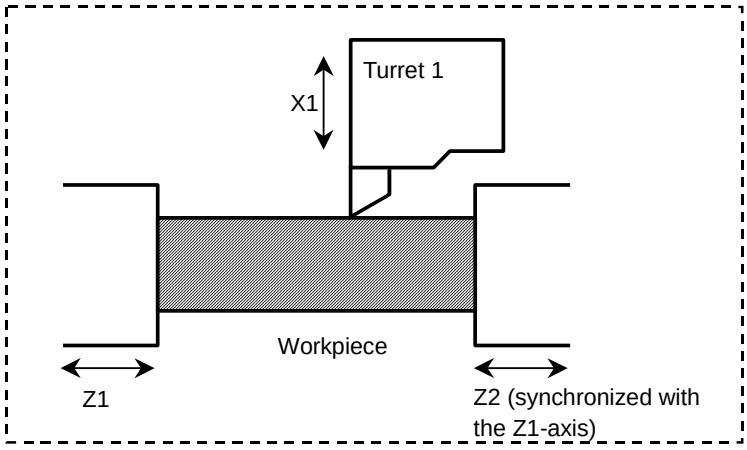


This function enables synchronous control between paths or within a path, composite control between paths, and superimposed control between paths, as explained below.

NOTE
Synchronous control and composite control is optional function.

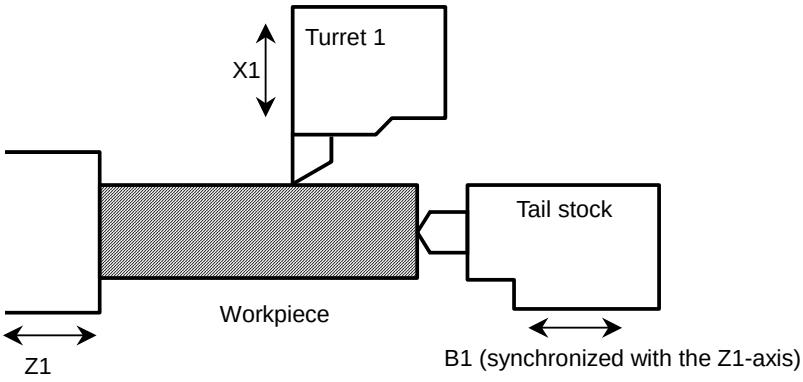
- Synchronous control

- Synchronization of an axis in one path with an axis in the other path
  (Example)
  Synchronization of the Z1-axis (master) with the Z2-axis (slave)



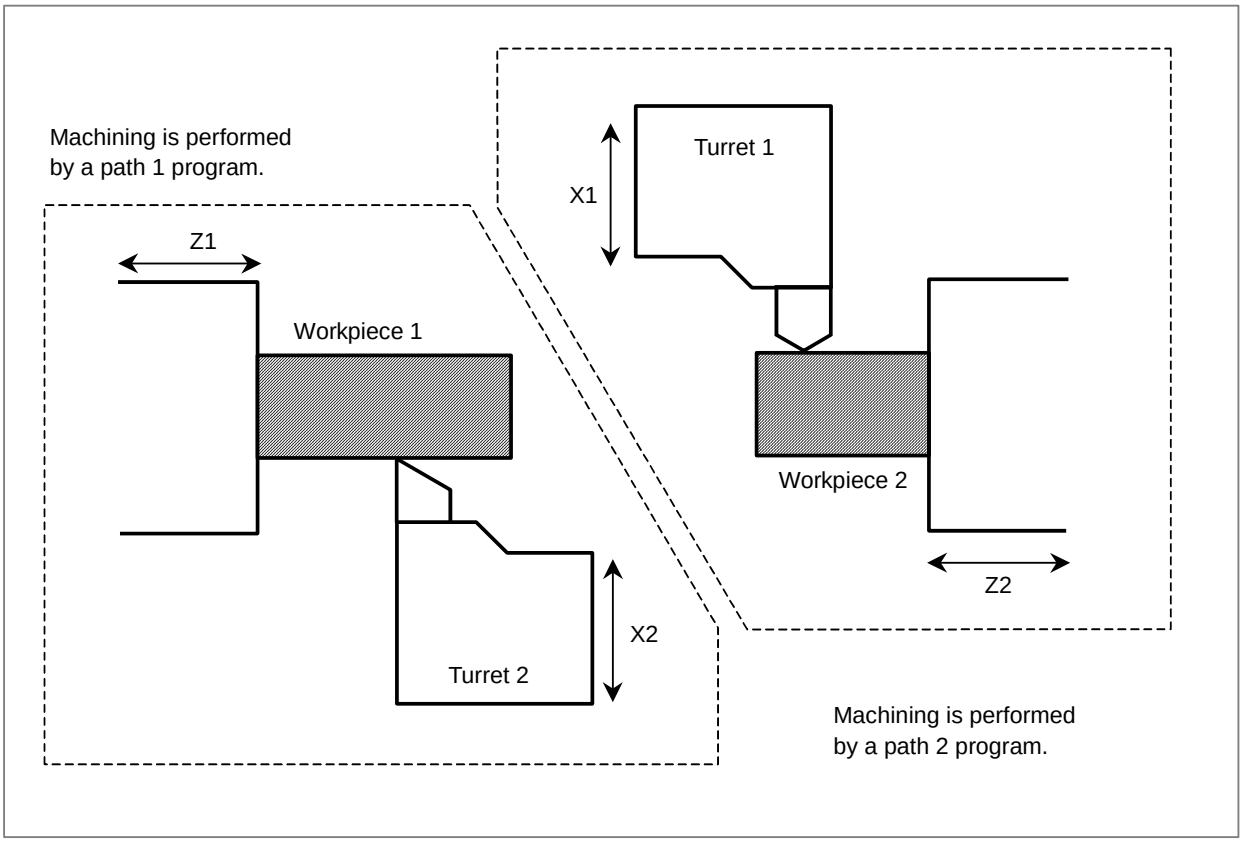
Machining is performed by a path 1 program.

- Synchronization of an axis in one path with another axis in the same path
  (Example)
  Synchronization of the Z1-axis (master) with the B1-axis (slave)



- Composite control

- Interchanging move commands for an axis in one path with those for an axis in the other path.
  (Example)
  Interchanging commands between the X1- and X2-axes
  → Control both X2- and Z1-axes by commands in a path 1 program
  Control both X1- and Z2-axes by commands in a path 2 program



8.5.1 Synchronous Control

An axis in one path can be synchronized with another axis in the same path or an axis in another path. This is done by issuing the same move commands for one axis (synchronous master axis) to another axis (synchronous slave axis). Using parameter SMRx (No. 8162#0) can cause the slave axis to move in the direction opposite to that of the master axis. It is possible to place either the master or slave axis in a parking state. The term parking here means to discontinue giving move commands to a servo system. No coordinates are updated in the parking state. Note however that absolute and relative coordinates can be updated using parameter PKUx (No. 8162#2).

Explanation

- Setting

Specify which axis is to be the synchronous master axis, with parameter No. 8180, using the path number and the axis number.

(Example)

For an axis configuration in which all paths consist of X, Z, and Y axes

To synchronize the Z2-axis (slave) with the Z1-axis (master):

Parameter No. 8180z of path 2 = 102

To synchronize the Y2-axis (slave) with the X1-axis (master):

Parameter No. 8180y of path 2 = 101

To synchronize the Y1-axis (slave) with the X1-axis (master):

Parameter No. 8180y of path 1 = 101

- Programming

Both before and after the M codes for a start and a cancellation of synchronous control, specify a waiting M code.

Master axis path	Slave axis path	
:	:	
M100;	M100;	Waiting
	M200;	Start of synchronous control
M101;	M101;	Waiting
:	:	
Independent operation	Independent operation	
:	:	
M100;	M100;	Waiting
	M201;	Cancellation of synchronous control
M101;	M101;	Waiting
:	:	

- Signal operation

When synchronization begins or ends (when an M code is issued, for example), the synchronous control axis selection signals SYNC1 to SYNC5 for the slave axis (from the PMC to the CNC) are changed from "0" to "1" (to begin synchronization) or from "1" to "0" (to terminate synchronization). To place an axis in a parking state, a parking signal PK1 to PK5 is set to "1" for the target axis.

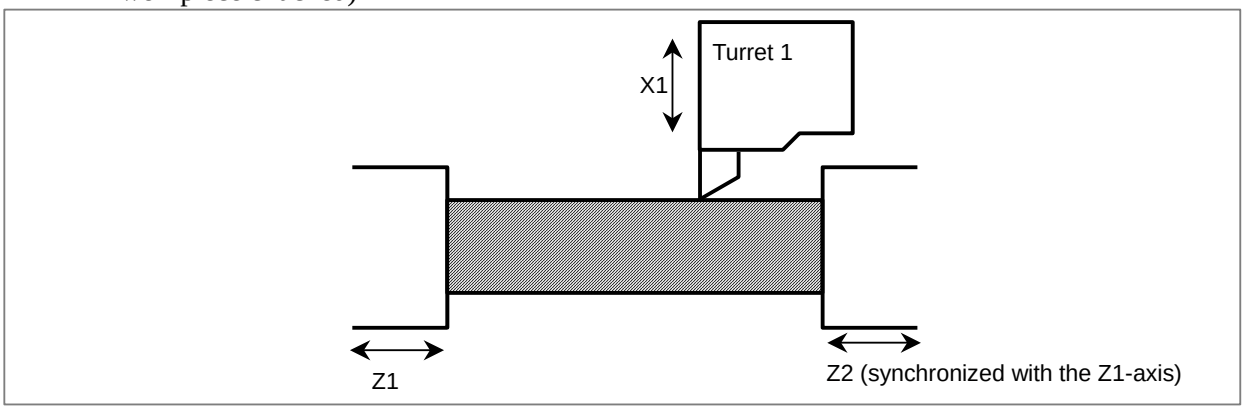
- Examples of applications

The following operations can be performed by using the synchronization functions together with the parking function, which causes move commands for an axis to be ignored and keeps the axis at a rest.

(1) Moving an axis in one path in synchronization with an axis in the other path (Both master and slave axes move.)

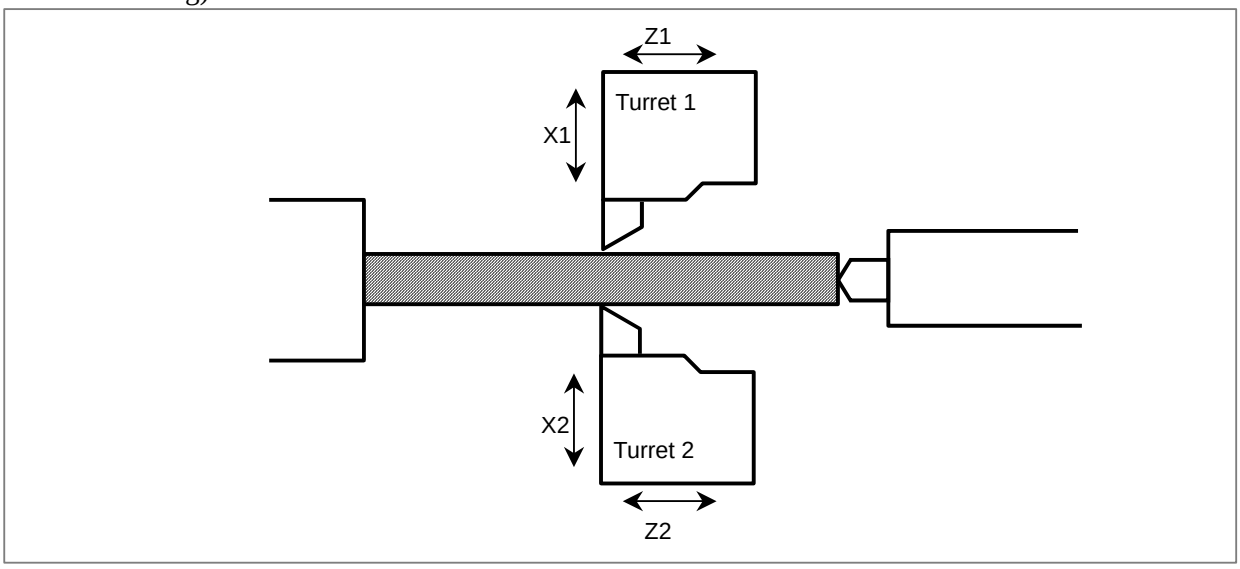
Example 1)

Synchronizing the Z2-axis (slave) with the Z1-axis (master) (machining with both ends of a workpiece chucked)



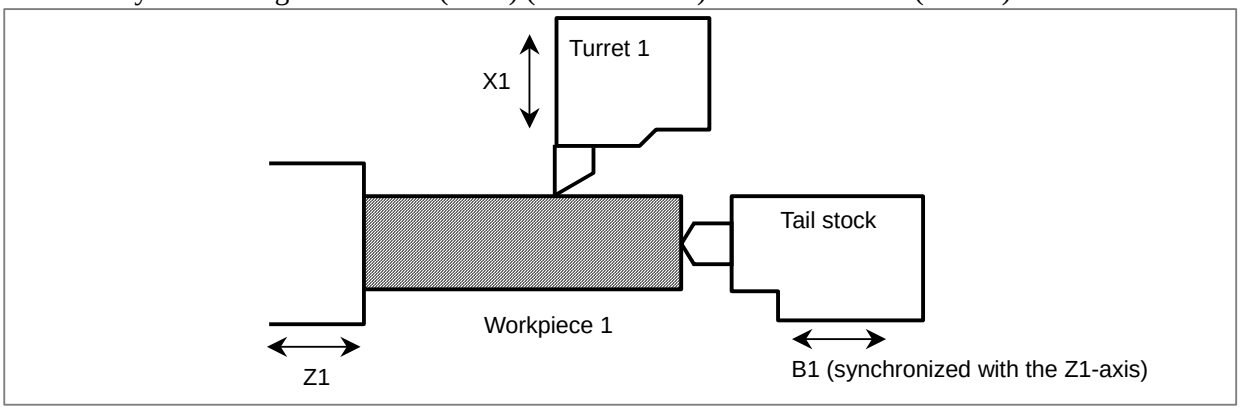
Example 2)

Synchronizing the X2- and Z2-axes (master) with the X1- and Z1-axes (slave) (balanced cutting)



Example 3)

Synchronizing the B1-axis (slave) (tail stock axis) with the Z1-axis (master)



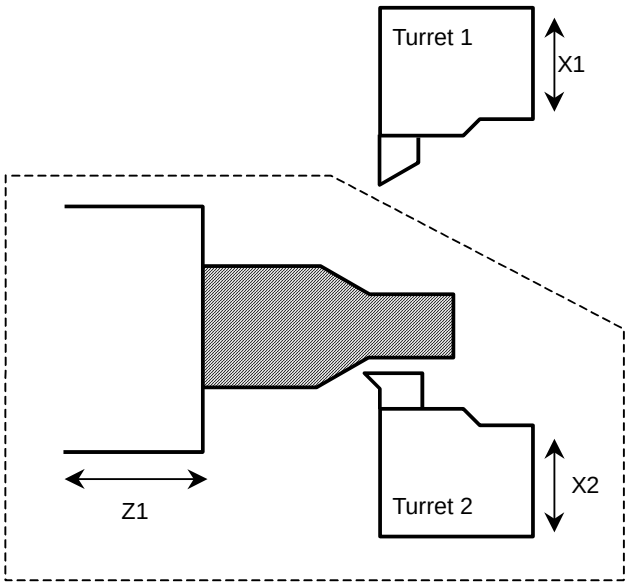
(2) Placing the movement along the synchronous master axis in the stopped state using a parking signal is referred to as master parking. In this state, the tool will move along the slave axes only. In contrast, placing the synchronous slave axes in the stopped state is referred to as slave parking. In this state, the tool will move along the master axis only. This makes it possible to control a single motor from both paths 1 and 2.

Example 4)

Sharing one motor with the Z1- and Z2-axes (assuming that the motor is linked to the Z1-axis)

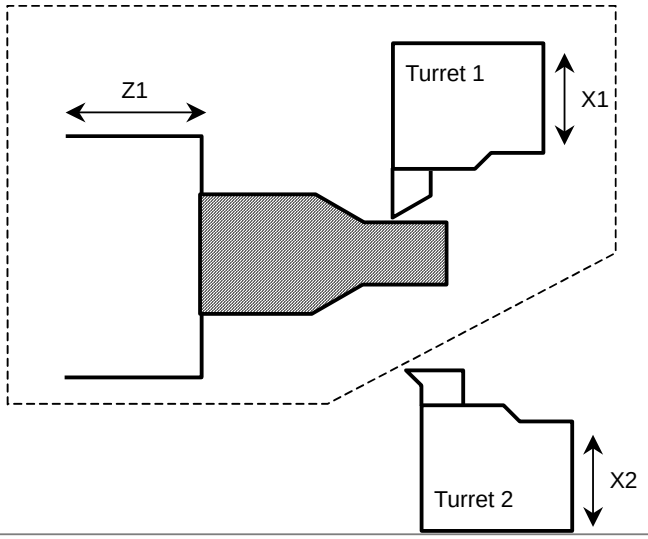
Master axis parking

The X2- and Z1-axes are moved by commands in a path 2 program (by synchronizing the Z1-axis (slave) with the Z2-axis (master)). At this point, the Z2-axis (master) is kept in a parking state.



Slave axis parking

The X1- and Z1-axes are moved by commands in a path 1 program (by synchronizing the Z2-axis (slave) with the Z1-axis (master)). At this point, the Z2-axis (slave) is kept in a parking state.



Because the coordinates of both Z1- and Z2-axes are updated, move commands can be executed immediately, without resetting up the coordinate system, when the synchronization state is switched.
Parking causes the positional relations between machine, absolute, and relative coordinates to shift. After reference position return, therefore, set the workpiece coordinate system.

- Reference position return and its check during synchronous control

If a reference position return command is issued for a synchronous master axis during synchronous control, it is executed normally for the master axis, but the slave axis does not return to its reference position (the slave axis only moves in synchronization with the reference position return of the master axis).
An exception is for automatic reference position return (G28) issued when the master axis is in a parking state, in which case the amount of movement is calculated so that the slave axis returns to its reference position provided that a reference position return has been carried for the slave axis. If the reference position of the slave axis has not been established, alarm (PS0354) will result.

If, in Cs axis synchronous control, the master axis is not in the parking state and the reference position of the master axis has not been established, a reference position return operation will be performed in reference to the master axis machine position, with the G28 command for the master axis.
If the master axis is in the parking state and the reference position of the slave axis has not been established, a reference position return operation will be performed in reference to the slave axis machine position, with the G28 command for the master axis.

If more than one slave axis belongs to one master axis, a reference position return command is executed so that the lowest-numbered slave axis returns to its reference position. If the master axis in one path is subjected to both synchronization with an axis in the same path and synchronization with an axis in the other path simultaneously, the lowest-numbered slave axis in the two paths is moved to the reference position.

A return to the second (third or fourth) reference position by the G30 command works in the same way as G28. In other words, usually only the master axis moves to the second (third or fourth) reference position. If the master axis is parking, the lowest-numbered axis is caused to move to its second (third or fourth) reference position.
If a reference position return check (G27) is specified during synchronous control, the master and slave axes move to the specified position. Upon completion of movement, a check is made to see whether the master axis is at its reference position (no check is made for the salve axes) unless the master axis is in a parking state, in which case a check is made upon completion of positioning to see whether the lowest-numbered slave axis is at its reference position.

- Out-of-synchronization detection

The term synchronous control used here only refers to an operation in which the same move command is issued to multiple different servo systems at one time. Note that synchronous control does not involve out-of-synchronization compensation, in which the positional deviation between multiple servo motors is constantly checked and one of the servo motors is subjected to compensation to reduce the deviation. However, using parameter SERx (No. 8162#1) can specify detection of out-of-synchronization.
Set the synchronization error limit in parameter No. 8181. If out-of-synchronization is detected, synchronization is canceled immediately and the servo ready state is turned off (alarm SV0407). The amount of synchronization error is shown in diagnosis No. 3502.

- Automatic setting of a workpiece coordinate system

When synchronous control is started in a workpiece coordinate system, it is possible to specify the workpiece coordinate system automatically. When synchronous control for a workpiece coordinate system is terminated, it is possible to return the workpiece coordinate system to ordinary machining (not synchronous control). The explanation of the workpiece coordinate system used during synchronous control follows. When synchronous control is used to move an axis differently from the way originally specified, for example, the master axis may be placed in a parking state, while the slave axis is allowed to move. In such a case, it will be convenient if a coordinate system that indicates the current position of the slave axis is used as a workpiece coordinate system for the master axis. Conventionally, this workpiece coordinate system must be specified by program when synchronous control is started, because the workpiece coordinate system does not originally belong to the master axis. This automatic workpiece coordinate system setting function for synchronous control sets up this workpiece coordinate system automatically.
This function can also resume the original workpiece coordinate system for the master axis automatically. Note that the workpiece coordinates of a salve axis can be automatically set through appropriate parameter setting at the end of synchronous control only.

- Setting and commands

In addition to setting ordinary synchronous control, parameters must be specified as follows:

(1) To set up a workpiece coordinate system for synchronous control automatically when starting synchronous control
Set parameter SPMx (No. 8163#1) to "1".
Set parameter No. 8185 with the coordinates of the slave axis reference position relative to the coordinates of the master axis when the master axis is at the reference position.

(2) To resume the ordinary workpiece coordinate system automatically when terminating synchronous control
Set parameter SPSx (No. 8163#2) to "1". (Only the parameter of the master axis.)
Set parameter No. 1250 with the master axis coordinates in the workpiece coordinate system when the master axis is at the reference position.
This synchronous control can be specified using the synchronous control axis selection signals (SYNC1 to SYNC5) similarly to the ordinary synchronous control. When the signals are raised to start synchronous control, a workpiece coordinate system for the master axis is automatically set up. When the signals are dropped to terminate synchronous control, the original workpiece coordinate system for the master axis is resumed automatically.

- Workpiece coordinate calculation method

(1) Workpiece coordinate system for synchronous control
(Master axis workpiece coordinate value)
= (parameter No. 8185 for the master axis)
± (slave axis machine coordinate value). . . <1>
+ (master axis machine coordinate value) . . . <2>
<1> +: Master axis parameter SCDx (No. 8163#4) = 0
-: Master axis parameter SCDx (No. 8163#4) = 1
<1> Master axis parameter SCMx (No. 8163#3) = 1 only

(2) Workpiece coordinate system for ordinary operation
(Master axis workpiece coordinate value)
=(parameter No. 1250 for the master axis)
+(master axis machine coordinate value)

Caution

⚠ CAUTION
The same least command and input increments must apply to both master and slave axes.

Note

NOTE
1 If more than one slave axis is synchronized with one master axis, the master axis is set with the workpiece coordinate system that corresponds to the current position of the first slave axis that is synchronized with the master axis.
2 A coordinate system can also be set with consideration given to tool offset, through appropriate parameter settings. So, the coordinate system is set up normally even when tool geometry compensation is applied.

8.5.2 Composite Control

Move commands can be interchanged between an axis in one path and an axis in the other path. In other words, when a machining program is executed for one path, actual machining can be performed with an axis in the other path. Coordinate systems can also be switched automatically between independent control and composite control.

Explanation

- Setting

Specify which axis is to interchange with which axis, with parameter No. 8183, using the path numbers and the axis numbers.

Example)

For an axis configuration in which all paths consist of X, Z, and Y axes

Between the Z1- and Z2-axes:

Parameter No. 8183z of path 2 = 102

Between the X1- and X2-axes:

Parameter No. 8183x of path 2 = 101

Between the X1- and Y2-axes:

Parameter No. 8183y of path 2 = 101

To set up coordinate systems automatically when composite control begins or ends, set parameters MPMx and MPSx (No. 8162 #4 and #5) to "1", and specify the positional relationship between the coordinate systems in parameter No. 8184.

- Programming

Both before and after the M codes for a start and a cancellation of composite control, specify a waiting M code.

Composite control source	Composite control destination	
:	:	
M100;	M100;	Waiting
	M210;	Start of composite control
M101;	M101;	Waiting
:	:	
Independent operation	Independent operation	
:	:	
M100;	M100;	Waiting
	M211;	Cancellation of composite control
M101;	M101;	Waiting
:	:	

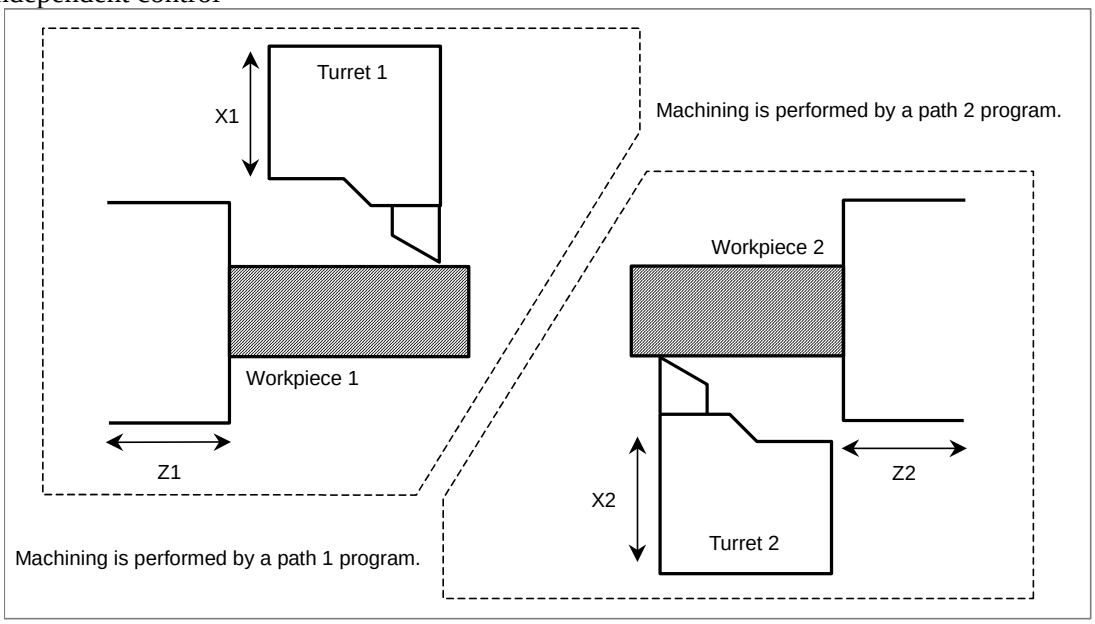
- Signal operation

When you want to start or cancel composite control (when you specify an M code, for example), change the composite control axis select signal between MIX1 and MIX5 for the axis subject to composite control that is specified with parameter No. 8183 (the signal being sent from the PMC to the CNC) from "0" to "1" (to start composite control) and from "1" to "0" (to cancel composite control).

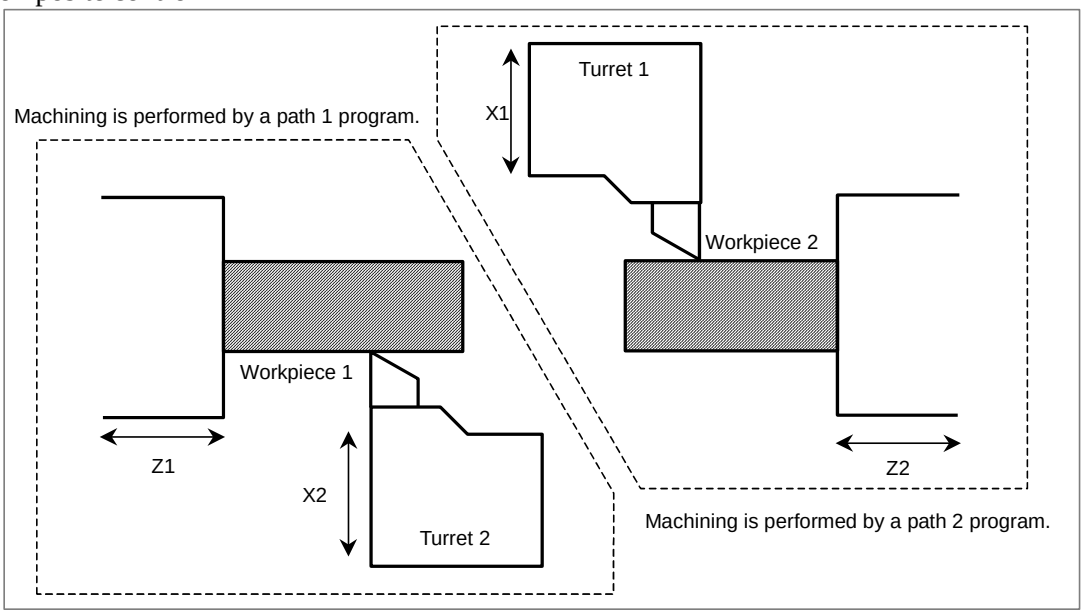
- Examples of applications

Suppose that a machine has the X1- and Z1-axes belonging to path 1 and the X2- and Z2-axes belonging to path 2 and that a workpiece moves along the Z1- and Z2-axes as directed by move commands. The following examples interchange commands between the X1- and X2-axes.

(1) Independent control



(2) Composite control



During composite control, the X2- and Z1-axes are moved by a path 1 program, and the workpiece coordinates of the X-axis in path 1 indicates the position of turret 2. Similarly, the X1- and Z2-axes are moved by a path 2 program, and the workpiece coordinates of the X-axis in path 2 indicates the position of turret 1.

- Spindle control

The composite control function does not switch the spindle speed command or the feed per rotation command based on feedback pulses from the position coder. Therefore, the spindle speed command and feedback pulses should be switched using the spindle command selection signals and spindle feedback selection signals.

- Tool offset during composite control

At the time of a switch between independent control and composite control, the preset offset value, tool nose radius compensation/cutter compensation/tool length amounts will not be changed. After a control switch, you must issue a new T/D/H code command to set an offset value again, if required.

- Reference position return during composite control

If G28 is issued to specify an automatic reference position return for an axis in one path during composite control, an amount of movement is calculated so that the associated axis in the other path can move to the reference position. In this case, the reference position for that axis must have already been established. If the reference position of the axis of the other path subject to composite control has not been established, alarm (PS0359) will result.
A manual reference position return is not allowed.
Note, however, that during Cs axis composite control, a manual reference position return is allowed. If, after the establishment of a reference position and before a switch to composite control, you want to enter the state in which no reference position is established, set parameter CRZ (No.8161#5) to 1.

- Automatic workpiece coordinate system setting

By parameter setting, a workpiece coordinate system to be used during composite control can be automatically set when composite control is started. When composite control is terminated, the workpiece coordinate system can be automatically returned to the workpiece coordinate system used for ordinary machining not under composite control.
A workpiece coordinate system to be used during composite control is explained below. For example, when a movement is made on axes other than the specified axes by using composite control, it is convenient to use a coordinate system that indicates the current position on the movement axes, as a workpiece coordinate system for the specified axes. Such a workpiece coordinate system differs from the original workpiece coordinate system for the specified axes, so that the coordinate system needs to be set by programming when composite control is started. Moreover, the workpiece coordinate system needs to be returned to the original workpiece coordinate system for the specified axes when composite control is terminated. With this function, these coordinate systems can be set automatically.

Setting, command

When composite control is started, a workpiece coordinate system to be used during composite control is set for axes with bit 4 (MPMx) of parameter No. 8162 set to 1. When composite control is terminated, the workpiece coordinate system can be automatically returned to the workpiece coordinate system used for ordinary machining not under composite control in connection with the axes with bit 5 (MPSx) of parameter No. 8162 set to 1.

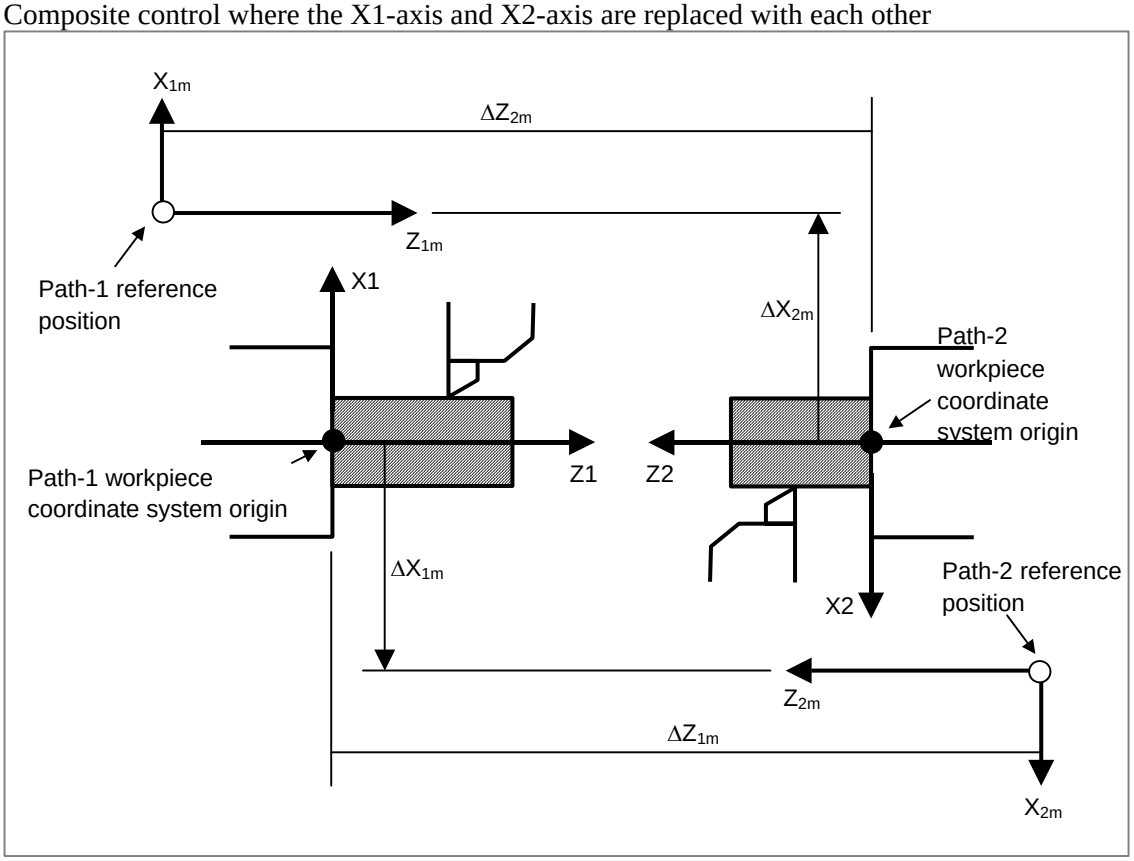
To specify this function, the composite control axis change selection signals MIX1 to MIX5 are used as in the case of ordinary composite control. When composite control is started by turning on signals, a workpiece coordinate system for the composite axes is automatically set. Similarly, when composite control is canceled by turning off signals, a workpiece coordinate system for the composite axes is automatically set.

Workpiece coordinate calculation method

1) When a workpiece coordinate system is not used (1 is set in bit 0 (NWZ) of parameter No. 8136), a coordinate value calculated from the coordinate value of the reference position of the mate axis in the local workpiece coordinate system (parameter No. 8184) and the machine coordinate value of the mate axis is set in a coordinate system at composite control start time.
   In a coordinate system at composite control termination time, a coordinate value calculated from the coordinate value (parameter No. 1250) of the reference position used for automatic coordinate system setting on the local axis and the machine coordinate value on the local axis is set.
   The method of coordinate value calculation is described below.

(Example)

Composite control where the X1-axis and X2-axis are replaced with each other



The reference position of path 2 in the workpiece coordinate system of path 1 is at (ΔX1m,ΔZ1m).
The reference position of path 1 in the workpiece coordinate system of path 2 is at (ΔX2m,ΔZ2m).
Set ΔX1m in parameter No. 8184x for path 1. Set ΔX2m in parameter No. 8184x for path 2.

When composite control is started, a workpiece coordinate system is set to satisfy the following:
X1 = (Path-1 X-axis setting) ± (X2 machine coordinate value)
+ when bit 6 (MCDx) of parameter No. 8162 for path 1 = 0
– when bit 6 (MCDx) of parameter No. 8162 for path 1 = 1
X2 = (Path 2 X-axis setting) ± (X1 machine coordinate value)
+ when bit 6 (MCDx) of parameter No. 8162 for path 2 = 0
– when bit 6 (MCDx) of parameter No. 8162 for path 2 = 1

When composite control is terminated, a workpiece coordinate system is set to satisfy the following:
X1 = (Parameter No. 1250 for path 1) + (X1 machine coordinate value)
X2 = (Parameter No. 1250 for path 2) + (X2 machine coordinate value)

2) When a workpiece coordinate system is used (0 is set in bit 0 (NWZ) of parameter No. 8136), a coordinate value is set through the workpiece coordinate system preset operation (equivalent to G92.1 IP 0) as described below, instead of being calculated with the method explained in 1) above.
When composite control is started, the workpiece coordinate system based on specified axes is preset to a workpiece coordinate system shifted on the specified axes by the currently valid workpiece origin offset values from the machine origin on the movement axes.
When composite control is terminated, the workpiece coordinate system based on composite axes is preset to a workpiece coordinate system shifted on the local axes by the currently valid workpiece origin offset values from the machine origin on the local axes.
However, when 1 is set in bit 7 (MWK) of parameter No.11284, the calculation method described to above 1) is applied even if the work coordinate system is used.

Signals

Synchronous control axis selection signals SYNC1 to SYNC5 <Gn138.0 to Gn138.4>

[Classification] Input signal

[Function] These signals perform synchronous control.

[Operation] When one of these signals becomes "1", the control unit:

Begins synchronous control in such a way that the corresponding axis becomes a slave axis.

The axis with which the slave axis is synchronized is determined by parameter No. 8180.

Parking signals PK1 to PK5 <Gn122.0 to Gn122.4>

[Classification] Input signal

[Function] These signals place each axis in a parking state.

[Operation] When one of these signals becomes "1", the control unit:

- Places the corresponding axis in a parking state.
  If the corresponding axis is under synchronous control, it enters a parking state immediately regardless of whether the axis is moving. If a parking signal is set to "1" without specifying synchronous control, it is ignored.

Composite control axis change selection signals MIX1 to MIX5 <Gn128.0 to Gn128.4>

[Classification] Input signal

[Function] These signals perform composite control.

[Operation] When one of these signals becomes "1", the control unit:

- Begins composite control over the corresponding axis.
  The axis with which the corresponding axis is controlled together is determined by parameter No. 8183.

NOTE
For parameter MIX (No.8166#1), only a single signal for path 1 is available.

Synchronous/composite/superimposed control under way signals SYN1O to SYN5O <Fn118.0 to Fn118.4>

[Classification] Output signal

[Function] These signals indicate each axis is being subjected to synchronous/composite/superimposed control.

[Output cond.] These signals become "1" under the following condition:

- The corresponding axis is under synchronous, composite, or superimposed control.

These signals become "0" under the following condition:

- The corresponding axis is not under synchronous, composite, or superimposed control.

> ⚠ **CAUTION**
> Whether each axis is under synchronous or composite control does not always match whether the corresponding selection signal (synchronous control axis selection or composite control axis selection) has been issued or not. For example, if these signals are set to "1" during an alarm, they are ignored. If a servo alarm occurs during these types of control, they are terminated automatically. Before attempting to perform these types of control, always check the state of these signals.

Synchronous master axis confirmation signals SYCM1 to SYCM5 <Fn341.0 to Fn341.4>

[Classification] Output signal
[Function] These signals notify whether the corresponding axes are synchronous master axes.
[Output cond.] These signals become "1" under the following condition:
- The corresponding axes are synchronous master axes.

These signals become "0" under the following condition:
- The corresponding axes are released from synchronous control.

Synchronous slave axis confirmation signals SYCS1 to SYCS5 <Fn342.0 to Fn342.4>

[Classification] Output signal
[Function] These signals notify whether the corresponding axes are synchronous slave axes.
[Output cond.] These signals become "1" under the following condition:
- The corresponding axes are synchronous slave axes.

These signals become "0" under the following condition:
- The corresponding axes are released from synchronous control.

Composite axis confirmation signals MIXO1 to MIXO5 <Fn343.0 to Fn343.4>

[Classification] Output signal
[Function] These signals notify whether the corresponding axes are composite control axes.
[Output cond.] These signals become "1" under the following condition:
- The corresponding axes are composite control axes.

These signals become "0" under the following condition:
- The corresponding axes are released from composite control.

Parking axis confirmation signals SMPK1 to SMPK5 <Fn346.0 to Fn346.4>

[Classification] Output signal
[Function] These signals notify whether the corresponding axes are parking axes in synchronous control.
[Output cond.] These signals become "1" under the following condition:
- The corresponding axes are parking axes in synchronous control.

These signals become "0" under the following condition:
- The corresponding axes are released from synchronous control or from parking.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn122				PK5	PK4	PK3	PK2	PK1
Gn128				MIX5	MIX4	MIX3	MIX2	MIX1
Gn138				SYNC5	SYNC4	SYNC3	SYNC2	SYNC1

	#7	#6	#5	#4	#3	#2	#1	#0
Fn118				SYN5O	SYN4O	SYN3O	SYN2O	SYN1O
Fn341				SYCM5	SYCM4	SYCM3	SYCM2	SYCM1
Fn342				SYCS5	SYCS4	SYCS3	SYCS2	SYCS1
Fn343				MIXO5	MIXO4	MIXO3	MIXO2	MIXO1
Fn346				SMPK5	SMPK4	SMPK3	SMPK2	SMPK1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8160	NRS	SPE	NCS					

[Input type] Parameter input
[Data type] Bit path

#5 NCS If an overtravel alarm is issued for an axis under synchronous, composite, or superimposed control, synchronous, composite, or superimposed control is:
0: Released.
1: Not released.

NOTE
If one of these paths is 1, both paths are assumed to be 1.

#6 SPE The synchronization deviation is:
0: The difference between the positioning deviation of the master axis and that of the slave axis.
1: The difference between the positioning deviation of the master axis and that of the slave axis plus the acceleration/deceleration delay.

NOTE
1 When the master and slave axes have different acceleration/deceleration time constants, set 1.
2 SPE is valid when bit 1 (SERx) of parameter No. 8162 is set to 1. SPE is used to find a synchronization deviation for comparison with parameter No. 8181.

#7 NRS When the system is reset, synchronous, composite, or superimposed control is:
0: Released.
1: Not released.

	#7	#6	#5	#4	#3	#2	#1	#0
8161	NSR		CRZ					NMR

[Input type] Parameter input
[Data type] Bit

#0 NMR When an axis subject to composite control is placed in servo-off state:
0: Composite control is stopped
1: Composite control is not stopped, provided bit 0 (FUP) of parameter No.1819 is set to 1 to disable follow-up for the axis.

#5 CRZ If the state of the composite control signal is switched in composite control on two axes under Cs contour control, the reference position establishment state of the two axes in composite control is:

0: Maintained. (The unestablished state is not assumed.)
1: Assumed to be unestablished.

#7 NSR When servo-off occurs with an axis in synchronous control:

0: Synchronous control is canceled.
1: Synchronous control is not canceled if follow-up operation is disabled for the axis (with bit 0 (FUPx) of parameter No. 1819 set to 1).

	#7	#6	#5	#4	#3	#2	#1	#0
8162	MUMx	MCDx	MPSx	MPMx		PKUx	SERx	SMRx

[Input type] Parameter input
[Data type] Bit axis

#0 SMRx Synchronous mirror-image control is:

0: Not applied. (The master and slave axes move in the same direction.)
1: Applied. (The master and slave axes move in opposite directions.)

#1 SERx The synchronization deviation is:

0: Not detected.
1: Detected.

NOTE
When both master and slave axes move in synchronization, the positioning deviations of the corresponding axes are compared with each other. If the difference is greater than or equal to the value specified in parameter No.8181, an alarm occurs. When either axis is in the parking or machine-locked state, however, the synchronization deviation is not detected.

#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 parameter SYWx (No. 8167#1)), set this parameter to 1.

#4 MPMx When composite control is started, the workpiece coordinate system is:

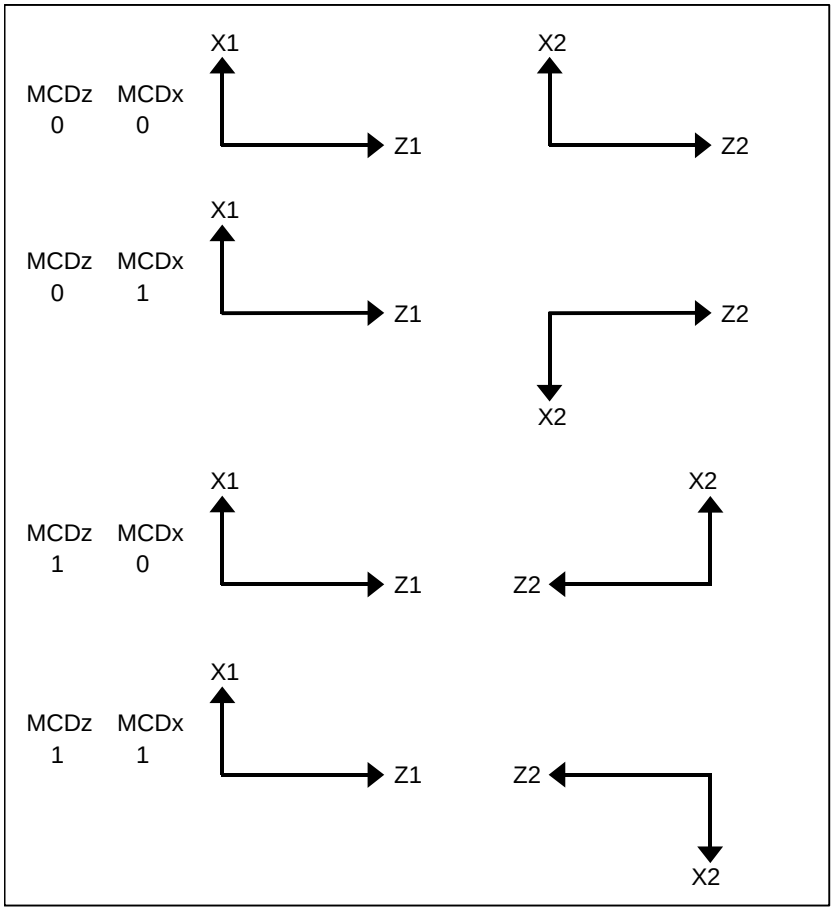
0: Not set automatically.
1: Set automatically.

NOTE
When the workpiece coordinate system is automatically set at the start of composite control, it is calculated from the following: Current machine coordinates and the workpiece coordinates at the reference point of each axis (parameter No.8184).
When the workpiece coordinate system is used (bit 0 (NWZ) of parameter No. 8136 is 0), however, instead of the coordinates obtained by the above calculation, the workpiece coordinates by the operation of workpiece coordinate system presetting (equivalent to G92.1 IP 0) in the machine coordinate system of the composite partner axis are set.

#5 MPSx When composite control is terminated, the workpiece coordinate system is:
0: Not set automatically.
1: Set automatically.

NOTE
When the workpiece coordinate system is automatically set at the end of composite control, it is calculated from the following: Current machine coordinates and the workpiece coordinates at the reference point of each axis under composite control (parameter No.1250)
When the workpiece coordinate system is used (bit 0 (NWZ) of parameter No. 8136 is 0), however, instead of the coordinates obtained by the above calculation, the workpiece coordinates by the operation of workpiece coordinate system presetting (equivalent to G92.1 IP 0) in the machine coordinate system of the local axis are set.

#6 MCDx The axes to be replaced with each other under composite control have the coordinate systems placed:
0: In the same direction. Simple composite control is applied. (A movement is made in the same direction along the corresponding axis.)
1: In opposite directions. Mirror-image composite control is applied. (A movement is made in the reverse direction along the corresponding axis.)



#7 MUMx In composite control, a move command for the axis:

0: Can be specified.
1: Cannot be specified.

NOTE
Upon the execution of a move command along an axis for which MUMx is set to 1 during composite control, alarm PS0353 is issued. For example, when axis X1 and axis X2 are placed under composite control, and a command for axis X2 (motor for axis X1) is to be disabled, set MUMx for path 2 to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
8163	NUMx	MMIx	SMIx	SCDx	SCMx	SPSx	SPMx	

[Input type] Parameter input
[Data type] Bit axis

#1 SPMx When synchronous control is started, automatic workpiece coordinate system setting for the master axis is

0: Not Performed.
1: Performed.

NOTE
When a workpiece coordinate system is automatically set at the start of synchronous control, the workpiece coordinate system is calculated from the current machine coordinates and the workpiece coordinates of each axis at the reference position set in parameter No.8185.

#2 SPSx When synchronous control terminates, automatic workpiece coordinate system setting for the master 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.

#3 SCMx When workpiece coordinates are calculated in synchronous control:
0: The workpiece coordinates are calculated from the machine coordinates of the slave axis.
1: The workpiece coordinates are calculated from the machine coordinates of the master axis and slave axis.

#4 SCDx The positive (+) directions of the master axis and slave axis in the coordinate system in synchronous control are:
0: Identical.
1: Opposite.
Set the parameters SPMx, SPSx, SCMx, and SCDx for the master axis. These settings are referenced during automatic workpiece coordinate setting for the master axis at the start of synchronous control.

#5 SMIx In synchronous control, the manual handle interruption amount for the master axis or the mirror image mode is:
0: Reflected in the slave axis.
1: Not reflected in the slave axis.

When this bit (SMIx) is set to 0
Manual handle interruption :
To the travel distance along the slave axis, the interruption amount of the master axis is also added.
Mirror image :
When mirror image is applied to the master axis, mirror image is also applied to the slave axis.

When this bit (SMIx) is set to 1
Manual handle interruption :
To the travel distance along the slave axis, the interruption amount of the master axis is not added.
Mirror image :
Even when mirror image is applied to the master axis, mirror image is not applied to the slave axis.

#6 MMIx For a composite control axis, manual handle interruption under composite control is:
0: Enabled.
1: Disabled.

#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 is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
8164					MCEx	MCSx	MWEx	MWSx

[Input type] Parameter input
[Data type] Bit axis

#0 MWSx In automatic workpiece coordinate system setting, performed when composite control is started, a workpiece shift and position offset are:
0: Not considered.
1: Considered.

NOTE
MWSx is valid when bit 4 (MPMx) of parameter No. 8162 is 1 and the workpiece coordinate system is not used (bit 0 (NWZ) of parameter No. 8136 is 1).

#1 MWEx In automatic workpiece coordinate system setting, performed when composite control is canceled, a workpiece shift and position offset are:
0: Not considered.
1: Considered.

NOTE
MWEx is valid when bit 5 (MPSx) of parameter No. 8162 is 1 and the workpiece coordinate system is not used (bit 0 (NWZ) of parameter No. 8136 is 1).

#2 MCSx For automatic setting of a workpiece coordinate system at composite control start time:
0: Parameter No. 8184 and the machine coordinate system of the composite control target path are used.
1: The absolute coordinate system of the composite control target path is used.

NOTE
MCSx is valid when bit 4 (MPMx) of parameter No. 8162 is 1 and the workpiece coordinate system is not used (bit 0 (NWZ) of parameter No. 8136 is 1).

#3 MCEx For automatic setting of a work coordinate system at composite control cancellation time:
0: Parameter No. 1250 and the machine coordinate system of the composite control target path are used.
1: The absolute coordinate system of the composite control target path is used.

NOTE
MCEx is valid when bit 5 (MPSx) of parameter No. 8162 is 1 and the workpiece coordinate system is not used (bit 0 (NWZ) of parameter No. 8136 is 1).

	#7	#6	#5	#4	#3	#2	#1	#0
8166							MIX	

[Input type] Parameter input
[Data type] Bit

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#1 MIX Composite control uses:
0: A new 2-path interface. In this case, the composite partner axis is set in parameter No. 8183 and the signal of the axis set in parameter No. 8183 is used as the composite control axis selection signal.
1: The conventional 2-path interface. In this case, parameter No. 8183 is set on path 2 and the signal on path 1 is used as the composite control axis selection signal.

	#7	#6	#5	#4	#3	#2	#1	#0
8167		SPVx	SWSx	SWMx	SGSx	SGMx	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		WST		MWR		SVF	MSO	MPA

[Input type] Parameter input
[Data type] Bit

#0 MPA When an alarm about synchronization/composite/superposition control is issued:
0: Both paths are set to the feed hold state.
1: Only the path that includes an axis related to synchronization/composite/superposition control is set to the feed hold state.
For example, in synchronization control within the same path, the path in which an alarm occurred is set to the feed hold state.

#1 MSO When one of the following events occurs in synchronous control or composite control:
- The emergency stop signal is turned off.
- The servo-off signal is turned on.
- A servo alarm is issued.

0: The synchronous or composite control mode is canceled and follow-up operation is not performed.
For the operation to be performed when the servo-off signal is turned on, however, the setting of bit 7 (NSR) of parameter No. 8161 is used in synchronous control or the setting of bit 0 (NMR) of parameter No. 8161 is used in composite control.

1: The synchronous or composite control mode is not canceled. The following operation is performed to perform follow-up operation:
When the emergency stop signal is turned off, if the other path is involved, the path operates so that the emergency stop signal is virtually turned off.
When the servo-off signal is turned on, the relevant axis is determined and operation is performed so that the servo-off signal is virtually turned on for the determined axis.
When a servo alarm is issued, the relevant axis is determined and the alarm SV0003, "CONTINUATION OF SYNCHRONOUS OR COMPOSITE CONTROL DISABLED" is issued for the determined axis to stop moving the tool along the axis.
When bit 2 (SVF) of parameter No. 8168 is set to 1, this servo-off specification follows the SVF setting.

NOTE
This setting is valid also during operation. For all axes placed under synchronous or composite control, the emergency stop signal is turned off, the servo-off signal is turned on, or a servo alarm is issued.

#2 SVF When an axis under composite control is placed in the servo-off state:
0: Composite control is canceled.
1: Composite control is not canceled.

Follow-up specification follows the setting of bit 0 (FUPx) of parameter No. 1819.
When bit 2 (SVF) of parameter No. 8168 is set to 1, bit 0 (NMR) of parameter No. 8161 is invalid. Bit 1 (MSO) of parameter No. 8168, specification for servo-off, is also invalid.

NOTE
If a composite control axis is placed in the servo-off state when stopped, set this parameter to 1.

#4 MWR When the workpiece coordinate system is automatically set if synchronization/composite/superposition control is started or ended, the tool compensation number is:
0: Canceled.
1: Not canceled.

NOTE
This parameter is valid when the workpiece coordinate system is enabled (bit 0 (NWZ) of parameter No. 8136 is 0).

#6 WST When a workpiece coordinate system is automatically set up for a slave axis at the end of synchronous control, workpiece coordinate system presetting is:
0: Not performed.
1: Performed.

NOTE
This parameter is valid when the workpiece coordinate system is enabled (bit 0 (NWZ) of parameter No. 8136 is 0) and bit 6 (SPV) of parameter No. 8167 is 1.

	#7	#6	#5	#4	#3	#2	#1	#0
8169						MRFx	MVMx	MDMx

[Input type] Parameter input
[Data type] Bit axis

#0 MDMx As machine coordinates in composite control:
0: Coordinates for the local path are displayed.
1: Coordinates for the other path in composite control are displayed.

#1 MVMx In composite control, machine coordinates (#5021 and above) to be read are:
0: Machine coordinates of the local path.
1: Machine coordinates of the other path in composite control.

#2 MRFx In composite control, the rapid traverse rate is:
0: The rapid traverse rate for the specified axis.
1: The rapid traverse rate for the axis along which a movement is made.

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, . . .)
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.

8181	Synchronization error limit of each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
When the synchronization deviation detected (SERx of Bit #1 parameter No.8162 is set to 1), this parameter specifies the limit of the difference between the positioning deviation of the slave axis and that of the master axis. Set this parameter to the slave axis.

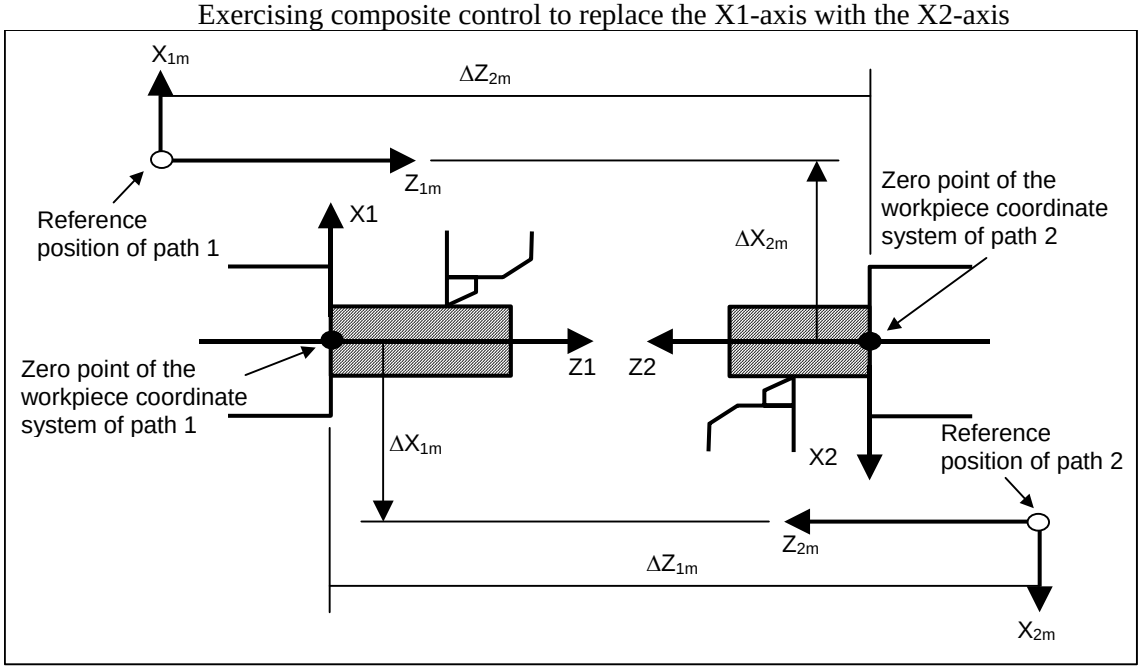
8183	Composite control axis of the other path in composite control for each axis

[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, . . .)
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 using the conventional 2-path interface (bit 1 (MIX) of parameter No. 8166 is 1, set this parameter on path 2. In this case, use the composite control axis selection signal on path 1.

8184	Coordinates of the reference point of an axis on the coordinate system of another axis under composite control

[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)
This parameter specifies the coordinates of the reference point of an axis on the coordinate system of another axis under composite control.
Example
Exercising composite control to replace the X1-axis with the X2-axis



(ΔX1m, ΔZ1m) are the coordinates of the reference point of path 2 on the workpiece coordinate system of path 1. (ΔX2m, ΔZ2m) are the coordinates of the reference point of path 1 on the workpiece coordinate system of path 2.

ΔX1m is specified for the parameter No. 8184x of path 1 and ΔX2m for the parameter No. 8184x of path 2.
If bit 4 (MPMx) of parameter No.8162 is set to 1 when composite control is started, the workpiece coordinate system satisfying the following conditions is specified:
X1 = (Value specified for the X-axis of path 1) ± (Machine coordinate of X2)
Plus when parameter MCDx (bit 6 of No.8162) of path 1 is set to 0
Minus when parameter MCDx (bit 6 of No.8162) of path 1 is set to 1
X2 = (Value specified for the X-axis of path 2) ± (Machine coordinate of X1)
Plus when parameter MCDx (bit 6 of No.8162) of path 2 is set to 0
Minus when parameter MCDx (bit 6 of No.8162) of path 2 is set to 1
If bit 5 of parameter No.8162 MPSx is set to 1 when composite control is terminated, the workpiece coordinate system satisfying the following conditions is specified:
X1 = (Parameter No.1250 of path 1) + (Machine coordinate of X1)
X2 = (Parameter No.1250 of path 2) + (Machine coordinate of X2)

NOTE
This parameter is valid when bit 4 (MPMx) of parameter No. 8162 is 1 and the workpiece coordinate system is not used (bit 0 (NWZ) of parameter No. 8136 is 1).

8185	Workpiece coordinates on each axis at the reference position

[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)
This parameter sets the workpiece coordinates on each master axis, subject to synchronous control, when the master and slave axes are at the reference position. This parameter is enabled when bit 1 (SPMx) of parameter No.8163 is set to 1. Set this parameter for the master axis.

	#7	#6	#5	#4	#3	#2	#1	#0
11284	MWK							

[Input type] Parameter input
[Data type] Bit

#7 MWK When the workpiece coordinate system is valid (when bit 0 (NWZ) of parameter No.8136 is 0), the behavior of the coordinate system setting when the composite control is begun or ended is:
0: The FS0i-D specification. (The preset which corresponds to G92.1 is performed.)
1: The FS0i-C specification.

Diagnosis screen

The diagnosis screen displays synchronous errors.

3502	Display of synchronous error for each axis

[Unit of data] Detection unit

[Meaning] Displays the difference between the master and slave axes in position deviation if synchronous deviation detection is performed (parameter No.8162 SERx=1).
The positional deviation difference is:
(positional deviation of the master axis) ± (positional deviation of the slave axis)

+ if a mirror image is applied to the synchronization command.
- if no mirror image is applied to the synchronization command.

Alarm and message

If any of the following alarms occurs, synchronous or composite control will be canceled for all axes.

- P/S alarm

Number	Message	Description
PS0350	PARAMETER OF THE INDEX OF THE SYNCHRONOUS CONTROL AXIS SET ERROR.	An illegal synchronization control axis number (parameter No. 8180) is set.
PS0351	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.
PS0352	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.
PS0353	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 parameter No. 8163#7NUMx 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 parameter No. 8163#7NUMx is set to 1. 2) A move command was issued to the axis for which parameter No. 8162#7MUMx is set to 1.
PS0354	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.
PS0355	PARAMETER OF THE INDEX OF THE COMPOSITE CONTROL AXIS SET ERROR.	An illegal composite control axis number (parameter No. 8183) is specified.
PS0356	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.
PS0357	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.
PS0359	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.

- D/S alarm

Number	Message	Description
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.

- Servo alarm

Number	Message	Description
SV0407	EXCESS ERROR	The difference value of the amount of positional deviation for the synchronization axis exceeded the setting value. (during synchronization control only)

Caution

- Items common to synchronous control and composite control

⚠ **CAUTION**

1 At the start or end of synchronous or composite control, the tool must be stopped on the axis subject to that control.
2 Before or after an M code for starting or canceling synchronous or composite control during automatic operation, be sure to specify a waiting M code (M code without buffering). When exercising synchronous or composite control in a path, be sure to prohibit look-ahead operation by specifying an M code without buffering before or after an M code for starting or canceling synchronous or composite control.
3 The axes under synchronous or composite control must match in least command increment, Detection unit, and diameter/radius specification. Otherwise, the amounts of travel will not be equal.
4 In synchronous or composite control, do not change the parameters related to it (including axis control, increment system, feedrate, and acceleration/deceleration control).
5 Before starting synchronous or composite control, make sure that the axes subject to it has undergone a reference position return after the power was turned on or that a reference position has been established with an absolute pulse coder.
6 If an emergency stop or servo off is performed or if a servo alarm occurs, the coordinates will change due to synchronous or composite control. If this occurs, "alarm DS1933 NEED REF RETURN (SYNC:MIX:OVL)" will be generated. After canceling the emergency stop, canceling servo off, or restoring from the servo alarm, perform a reference position return and coordinate system setting first, then start synchronous or composite control.
7 Acceleration/deceleration control, pitch error compensation, backlash compensation, and stored stroke check are performed independently, regardless of synchronous or composite control.
8 The following servo software functions cannot be used with synchronous or composite control:
   - Abnormal load detection and switching function for each of cutting and rapid traverse
   - Electric current 1/2 and PI switching function for each of cutting and rapid traverse
   - Torque command filter switching function for each of cutting and rapid traverse

⚠ CAUTION

9 The following functions cannot be used in synchronous or composite control:
- Spindle positioning

10 During synchronous or composite control, reference position establishment based on "linear scale with absolute address reference mark" or "linear scale with absolute address zero point" is impossible.

NOTE

1 You can place more than one axis under synchronous or composite control at the same time. You cannot, however, synchronize an axis to multiple axes at the same time, synchronize the interchanging axis under composite control to another axis, or duplicate an interchange.
2 Synchronous or composite control cannot be performed on a linear axis and a rotation axis.

- Items related to synchronous control only

⚠ CAUTION

1 In synchronous control, you cannot issue a move command for the synchronous axis on the synchronous slave.
2 The axes under synchronous control must match in acceleration/deceleration time constant and servo parameters, insofar as possible. If their settings greatly differ, the actual machine movement will deviate.
3 Even if you perform operations that do not move the machine but change the workpiece coordinate system only, such as workpiece coordinate system setting/shifting and geometric offset commands, on the synchronous master, this will not be reflected in the workpiece coordinate system on the slave.
4 If you perform workpiece coordinate system setting/wear offset commands, tool nose radius compensation, or cutter compensation on the synchronous master, the travel path on the synchronous slave will shift by the offset, but the offset will not be set as an offset amount. (No offset vector will be created.)
5 The synchronous or composite control and axis synchronous control can be specified at the same time. However, the master and slave axes used for axis synchronous control cannot be used for synchronous control.

- Items related to composite control only

⚠ CAUTION

1 If you want to place a tapping axis under composite control, place it under composite control first, then issue a rigid tapping command. In rigid tapping mode, you must not switch the composite control axis selection signal; be sure to do this in the rigid tapping canceled state.
2 If you place a tapping axis under composite control, rigid tapping will be performed using the time constant, loop gain, in-position width, and the positional deviation limits during stoppage/travel of the interchanging axis in composite control.
3 In composite control, the loop gain may be switched depending on the switching of the Cs contour control switching signal and of the composite control axis selection signal. For this reason, switch the signals in the state in which the tool has stopped on the related axis. If you change the loop gain while the tool is moving along an axis, shock may result.

⚠ CAUTION

4 The synchronous or composite control and axis synchronous control can be specified at the same time. An axis used for movement under composite control can be selected as the master axis for axis synchronous control, but cannot be specified as a slave axis for axis synchronous control.

Limitation

- Limitations on synchronous control and composite control

Function	In synchronous control	In composite control
Acceleration/deceleration control	The synchronous slave axis is subject to acceleration/deceleration of the same type as the synchronous master axis. As the time constants, those specific to the axes are used.	The acceleration/deceleration type of the specified path is used. As the time constants, those specific to the axes are used (3).
Acceleration/deceleration before interpolation	The synchronous slave axis is subject to the same acceleration/deceleration as the synchronous master axis.	The acceleration/deceleration settings of the specified path are used.
Cutting feedrate clamp	Clamping is performed on the synchronous master.	Clamping is performed on the specified path (4).
Reference position return	Possible on the synchronous master axis if the axis is not in parking. Only automatic reference position return (G28) is possible if the synchronous master axis is in parking.	Possible on axes not related to composite control. Only automatic reference position return (G28) is possible on an axis under composite control.
2nd, 3rd, 4th reference position return	Same as above.	Same as above.
Reference position return check	Same as above.	Same as above.
PMC axis control	Possible on axes other than the synchronous slave axis.	Possible.
Polar coordinate interpolation, Cylindrical interpolation	Possible.	Possible. (In polar coordinate interpolation or cylindrical interpolation mode, composite control cannot be turned ON/OFF.)
Handle interruption	See parameter SMI (No. 8163#5).	Possible with parameter MMI (No. 8163#6).
Mirror image	See parameter SMI SMI (No. 8163#5).	Signals in the specified path are effective (4).
Machine lock	Signals in the individual paths are effective (1).	Signals in the specified path are effective (4).
Inter lock	Signals on the synchronous master side are valid for the synchronous slave axis.(2)	Signals in the specified path are effective (4).
Override	Signals on the synchronous master side are valid for the synchronous slave axis.(2)	Signals in the specified path are effective (4).
External deceleration	Signals on the synchronous master side are valid for the synchronous slave axis.(2)	Signals in the specified path are effective (4).
Skip function	Not possible on the synchronous slave axis.	Possible on axes not related to composite control.
Automatic tool offset	Not possible on the synchronous slave axis.	Possible on axes not related to composite control.
Follow-up	Not possible during synchronization.	Not possible during composite control.

Function	In synchronous control	In composite control
Program restart	Not possible with a program containing synchronous control.	Not possible with a program containing composite control.
Cs contour control	Synchronous control possible (5).	Composite control possible (5).
Spindle positioning	Synchronous control does not possible	Composite control does not possible
EGB function	Synchronous control does not possible	Composite control does not possible
Servo off	Synchronous control does not possible	Possible unless the interchanging axis in composite control is in the servo off state.

(1) Processed after synchronous pulses are transferred to the slave.
(2) Processed on the master, then transfers synchronous pulses.
(3) Transfers composite control pulses and the acceleration/ deceleration type. As the time constant, uses that on the slave.
(4) Processed on the master, then transfers composite control pulses.
(5) Limited to combinations of Cs axes.

- Reading of coordinates in synchronous or composite control

In synchronous or composite control, the reading of custom macro system variable positional information or of current coordinates from the PMC window is as follows:

Positional information type	In synchronous control	In composite control
Absolute coordinate	Reading possible.	Reading possible.
Machine coordinate	Reading possible.	Reading possible.
End point of each block	Reading possible on the master only.	Reading possible.
Skip signal position	Reading possible on the master only.	Reading not possible.

- Canceling synchronous or composite control

Synchronous or composite control is canceled in the event of the following, as well as when the synchronous/composite control axis selection signal becomes "0".

(1) Emergency stop
(2) Reset
(3) Servo alarm
(4) Servo off *1
(5) Overtravel
(6) Alarm related to synchronous or composite control
(7) Alarm PW0000

If one of the above events occurs in either path, all paths will be released from synchronous or composite control. If one of the above events occurs in either path in synchronous or composite control, the other path will automatically be placed in feedhold state (during automatic operation) or in interlock state (during manual operation).

*1: By setting parameter NMR (No. 8161#0) to 1, you can prohibit the composite control axis from release from the synchronous or composite control state even if the axis enters the servo off state.
By setting parameter NSR (No. 8161#7) to 1, you can prohibit the synchronous control axis from release from the synchronous or composite control even if the axis enters the servo off state.

- Axis state output signals in synchronous or composite control

State output signal	In synchronous control	In composite control
Axis moving signals MVn <Fn102>	- During movement along the master or slave axis, the master axis signal goes "1". - The slave axis signal is always "0" (1).	- On the axis on which a move command is executed, the signal goes "1". The signal on the axis along which the tool is actually moving does not turn to "1" (1).

State output signal	In synchronous control	In composite control
Axis moving direction signals MVDn <Fn106>	- On the master axis, the moving direction of the master axis. - On the slave axis, moving direction after mirror image processing.	- Actual moving direction on the axis (moving direction after mirror image processing in composite control).
In-position signals INPn <Fn104>	- The master axis signal goes "1" when both the master and slave axes are in the in-position state. - The slave axis signal is always "1".	- The signal in the specified path reflects the state of the moving axis with that specification.
Reference position establishment signal ZRFn <Fn120>	- The signal on the axis for which the reference position has been established turns to "1".	- The signal on the axis for which the reference position has been established turns to "1".
Reference position return completion signal ZPn <Fn094>	- After completion of reference position return of the master axis, the signal on the master axis turns to "1". - The slave axis is synchronized with movement of the master axis, but is not subjected to reference position return. The signal turns to "0". - When the master axis is parking, the slave axis is subjected to reference position return. The signal on the slave axis turns to “1”.	- The signal on the axis along which the tool is actually moving turns to "1".

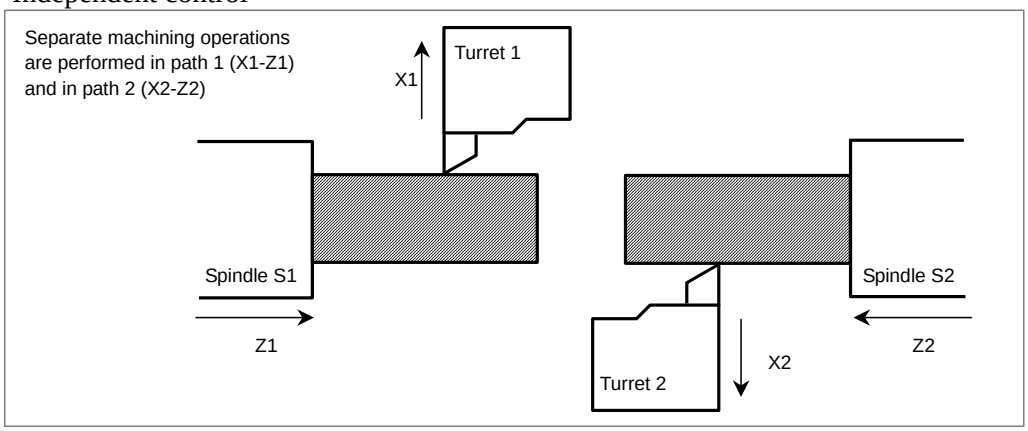
(1) In a positional deviation check, regardless of the states of these signals, parameter No. 1828 is used if move command pulses are sent to the motor (regardless of whether the axis is the master or slave) as the limit, and parameter No. 1829 if none are sent.

Examples of Use

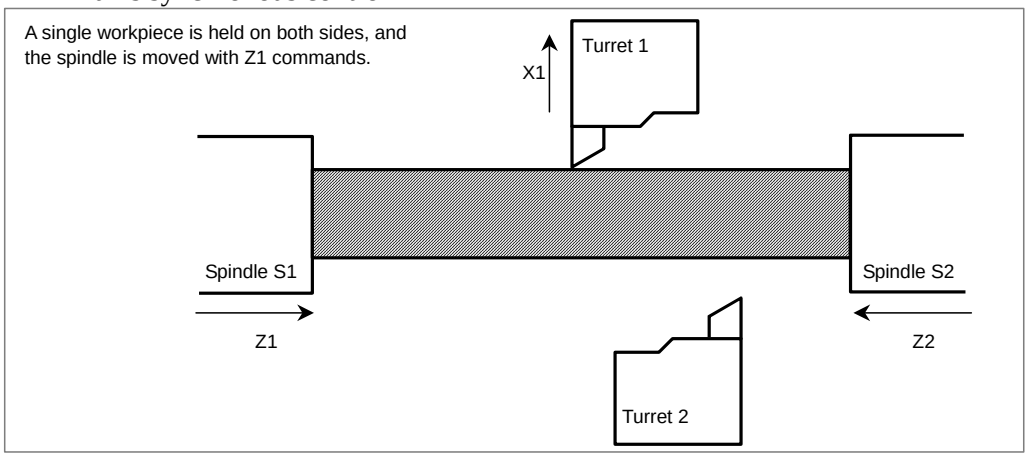
- Examples of independent control and of synchronous control on the Z1 and Z2 axes

(1) Machine configuration

(a) Independent control



(b) Z1-Z2 axis synchronous control



(2) Parameter settings

- To synchronize the Z2 (slave) axis with the Z1 (master) axis, set parameter No. 8180z of path 2 to "102".
- Because the plus directions of the Z1 (master) and Z2 (slave) axes are opposite, perform synchronization with a mirror image applied. To that end, set parameter SMRz (No.8162#0) of path 2 to "1".
- Because the movements along the Z1 (master) and Z2 (slave) axes must be equal, perform synchronous error detection. Set parameter SERz (No.8162#1) of path 2 to "1". Set the synchronous error limit in parameter No. 8181z of path 2, in the range of 100 to 1000. (The setting differs with each machine.)
- During synchronization, the difference between the Z1 (master) axis and the Z2 (slave) axis in positional deviation is displayed as No. 3502z on the diagnosis screen of path 2.

(3) Signal operation

- To start synchronous movement along the Z1 (master) axis and the Z2 (slave) axis, set signal SYNC2 <G1138.1> to "1".
- To cancel synchronization, set signal SYNC2 <G1138.1> to "0".
- When an emergency stop, NC reset, or alarm occurs, set signal SYNC2 <G1138.1> to "0".
- Set signals other than signal SYNC2 <G1138.1> to "0".

(4) Program example

Path 1		Path 2		
N1000	. . .	N2000	. . .	Independent machining of each path
N1010	Z80. ;	N2010	Z150. ;	Move the workpiece and the chuck to position
N1020	M200 ;	N2020	M200 ;	Waiting
		N2030	M61 ;	Workpiece clamping and start of synchronization
N1040	M201 ;	N2040	M201 ;	Waiting
N1050	M3 S800 ;			Forward spindle rotation
N1060	Z- 25. ;			Movement along the Z1 axis
N1070	. . .			
	Machining along the X1 and Z1 axes			
N1080	M200 ;	N2080	M200 ;	Waiting
		N1090	M62 ;	Cancellation of synchronization and unclamping of workpiece
N1100	M201;	N2100	M201;	Waiting
N1110	. . .	N2110	. . .	Independent machining of each path

where

M61 is an M code that clamps the workpiece and sets signal SYNC2 <G1138.1> to "1", and
M62 is an M code that sets signal SYNC2 <G1138.1> to "0" and unclamps the workpiece.

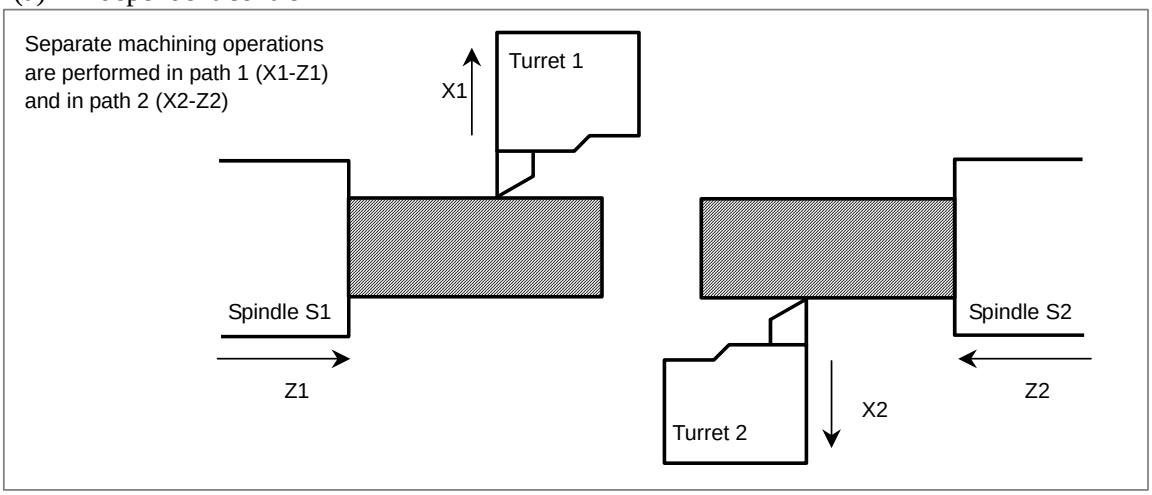
(5) Note

NOTE
1 An operation to make the speeds of spindles S1 and S2 identical is required. To do this, issue a spindle command of path 1 to both S1 and S2.
2 A waiting M code is an M code without buffering.

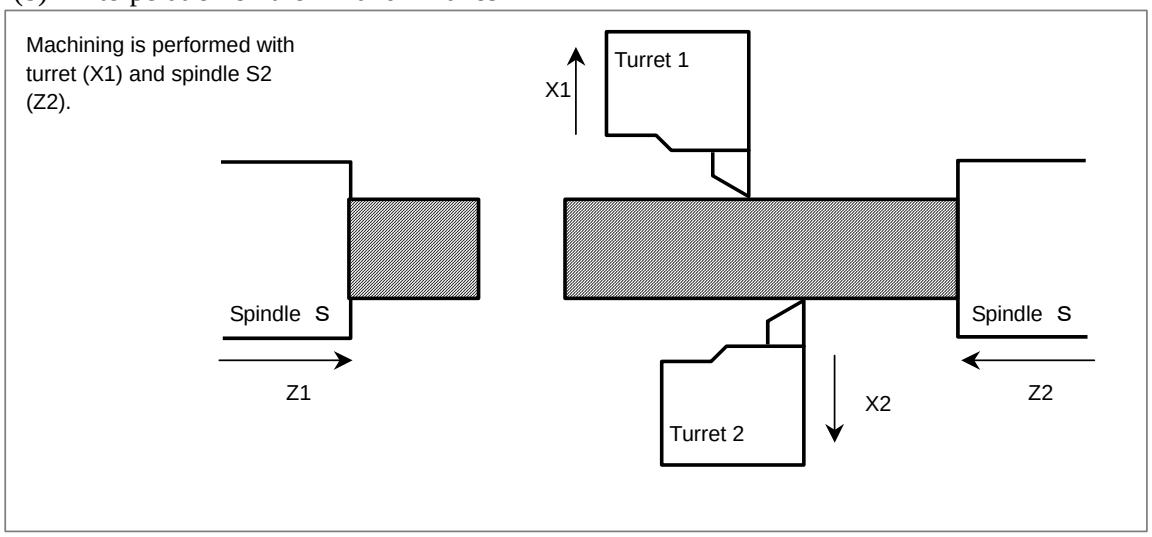
- Examples of independent control and interpolation on the X1 and Z2 axes

(1) Machine configuration

(a) Independent control



(b) Interpolation on the X1 and Z2 axes



There are two ways to perform interpolation on the X1 and Z2 axes.

1. Use the program of path 2 to issue a command to the X2 and Z2 axes to synchronize the X1 (slave) axis to the X2 (master) axis and to park the X2 (master) axis. For path 1, do not issue a move command.
2. Perform composite control to let the X1 and X2 axes interchange between paths. For path 1, do not issue a move command.

The following explains the ways using synchronous control and composite control separately.

If using synchronous control

(1) Parameter settings

- To synchronize the X1 (slave) axis to the X2 (master) axis, set parameter No. 8180x of path 1 to "201".
- For both X1 (slave) and X2 (master) axes, no mirror image is required because the direction away from the center of the workpiece is the plus direction of coordinates.
- Do not perform synchronous error detection because the X2 (master) axis is parked.
- During synchronization, the difference between the X2 (master) and X1 (slave) axes in positional deviation is displayed as No. 3502x on the diagnosis screen of path 1.

(2) Signal operation

- To start synchronous control on the X2 (master) axis and the X1 (slave) axis, set signals SYNC1 <G0138.0> and PK1 <G1122.0> to "1".
- To cancel synchronization, set signals SYNC1 <G0138.0> and PK1 <G1122.0> to "0".
- When an emergency stop, NC reset, or alarm occurs, set SYNC1 <G0138.0> and PK1 <G1122.0> to "0".
- Set signals other than SYNC1 <G0138.0> and PK1 <G1122.0> to "0".

(3) Program example

Path 1		Path 2		
N1000	. . .	N2000	. . .	Independent machining of each path
N1010	Z0 ;	N2010	Z20. ;	Move each workpiece to position
N1020	X120. ;	N2020	X120. ;	Move each X axis to the synchronization start position (X1=X2)
N1030	M200;	N2030	M200;	Waiting
		N2040	M55 ;	Synchronize the X2 and X1 axes and start parking on the X2 axis
N1050	M201 ;	N2050	M201;	Waiting
		N2060	T0212 ;	Set the offset for turret 1
		N2070	S1000 M4 ;	Reverse spindle rotation
		N2080	G0 X30. Z55. ;	
		N2090	G1 F0.2 W-15. ;	Machining along the X1 and Z2 axes
		N2100	. . .	
N1110	M200 ;	N2110	M200;	Waiting
		N2120	M56 ;	Cancellation of synchronization and parking
N1130	M201 ;	N2130	M201;	Waiting
N1140	. . .	N2140	. . .	Independent machining of each path

where

M55 is an M code to start the control of turret 1 with the program of path 2, and M56 is an M code that cancel the control of turret 1 with the program of path 2.

NOTE

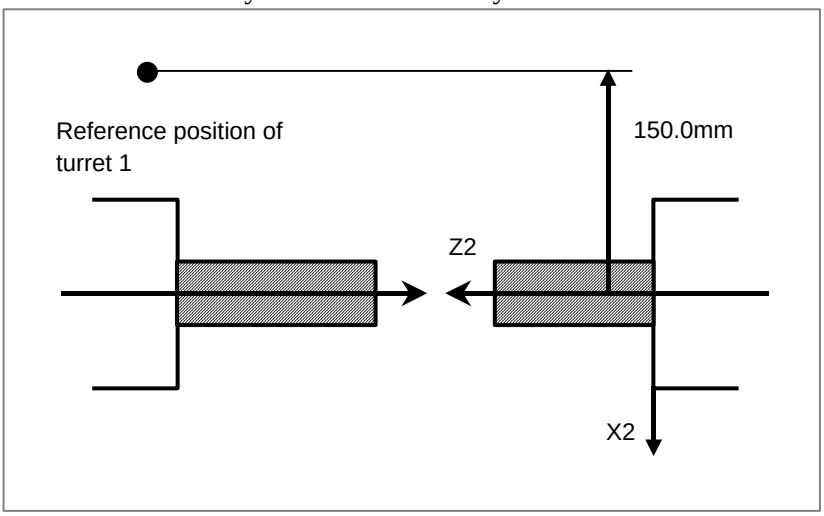
During synchronous control on the X axis, no move command can be issued from path 1 to the X1 (slave) axis, but movement along the Z1 axis is possible.

If using composite control

(1) Parameter settings

- To perform composite control by letting the X1 and X2 axes interchange, set parameter No.8183x of path 2 to "101".
- Because the direction of the coordinates on the X1 axis is opposite from that of the coordinates on the X2 axis, set parameter MCDx (No.8162#6) of path 2 to "1".
- To automatically set the position of turret 1 in the workpiece coordinate system of path 2 at the start of composite control, set parameter MPSx (No.8162#5) of path 2 to "1".
- To automatically set the position of turret 1 in the workpiece coordinate system of path 1 at the end of composite control, set parameter MPMx (No.8162#4) of path 1 to "1".

- Assuming that the X coordinate of the reference position of turret 1 in the workpiece coordinate system of path 2 is -150.0 mm as shown in the figure below, set parameter No. 8184x of path 2 to "-150000" to automatically set the coordinate system.



(2) Signal operation

- To start composite control on the X2 and X1 axes, set signal MIX1 <G1128.0> to "1".
- To cancel composite control, set signal MIX1 <G1128.0> to "0".
- When an emergency stop, NC reset, or alarm occurs, set MIX1 <G1128.0> to "0".
- Set signals other than MIX1 <G1128.0> to "0".

(3) Program example

Path 1		Path 2		
N1000	. . .	N2000	. . .	Independent machining of each path
N1010	Z0 ;	N2010	Z20. ;	Move each workpiece to position
		N2020	X120. ;	Move along the X2 axis to a non-interference position
N1030	M200 ;	N2030	M200 ;	Waiting
		N2040	M55 ;	Start composite control on the X2 and X1 axes (perform automatic coordinate system setup)
N1050	M201 ;	N2050	M201 ;	Waiting
		N2060	T0212 ;	Set the offset for turret 1
		N2070	S1000 M4 ;	
		N2080 G0 U10. W- 20. ;		Machining along the X1 and Z2 axes
		N2090 G1 F0.2 W- 15. ;		
		N2100	. . .	
N1110	M200 ;	N2110	M200 ;	Waiting
		N2120	M56 ;	Cancel composite control (perform automatic coordinate system setup)
N1120	M201 ;	N2120	M201 ;	Waiting
N1130	. . .	N2130	. . .	Independent machining of each path

where

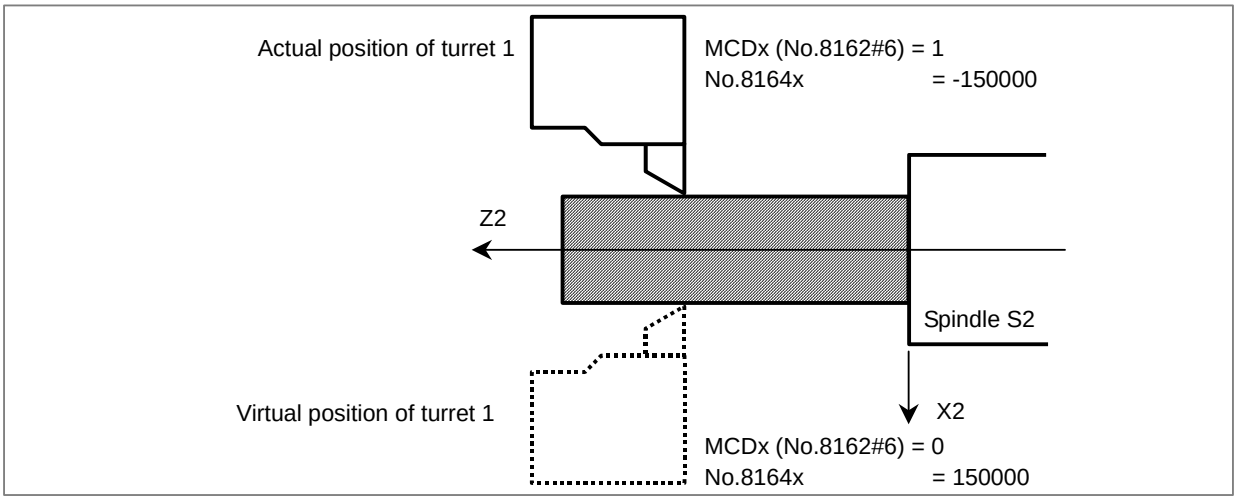
M55 is an M code to start the control of turret 1 with the program of path 2, and
M56 is an M code that cancel the control of turret 1 with the program of path 2.

NOTE

1 At the start and end of composite control, you do not necessarily perform automatic setup of the coordinate system. If you do not perform automatic setting, the program will set an appropriate one internally.
2 During composite control on the X axis, you can issue a move command for the X axis in path 1 to move the tool along the X2 axis.

NOTE

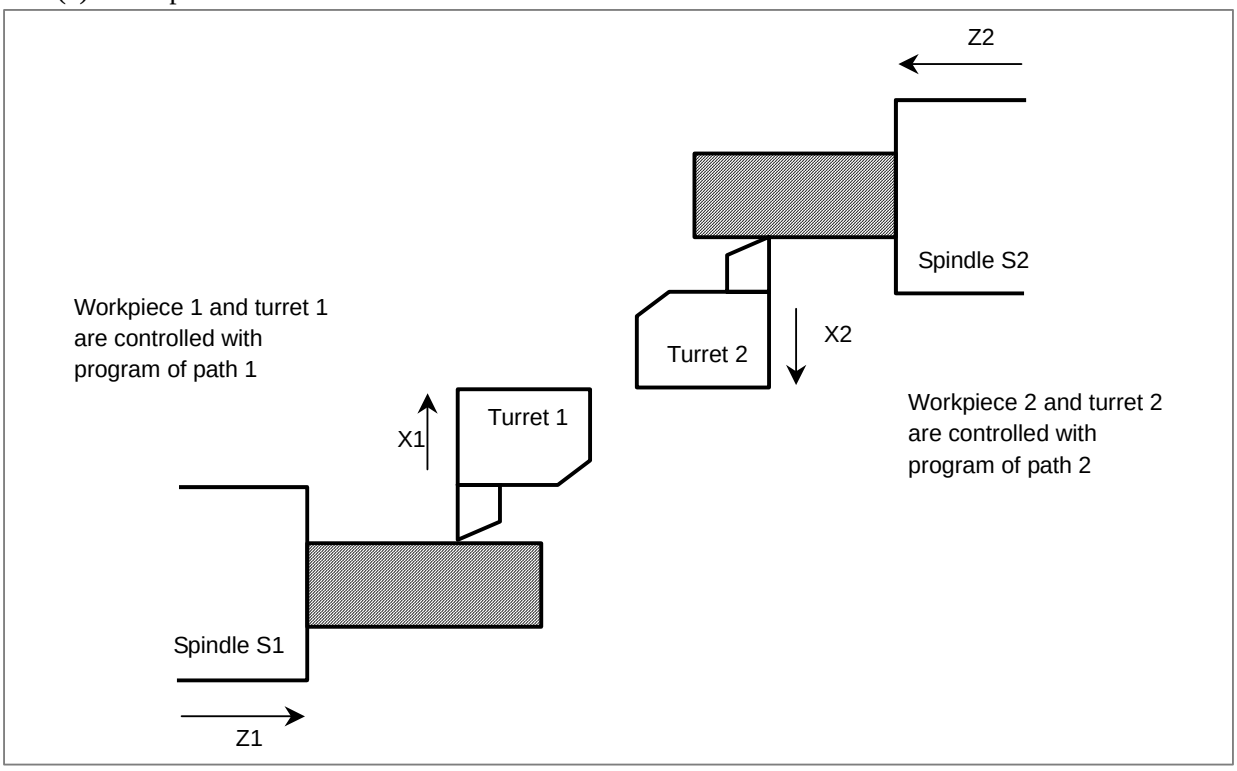
3 With the parameter settings above, turret 1 will be positioned on the minus side of the X coordinates in the workpiece coordinate system of path 2. For this reason, specify with the opposite sign as usual, such as U+10. and U-10. to move turret 1 toward the center of the workpiece and move it away from it, respectively.
If this is inconvenient, specify the following parameter settings:
Parameter MCDx (No.8162#6) = 0
Parameter No. 8184x = 150000
These settings cause turret 1 to exist on the plus side of the X coordinates virtually.



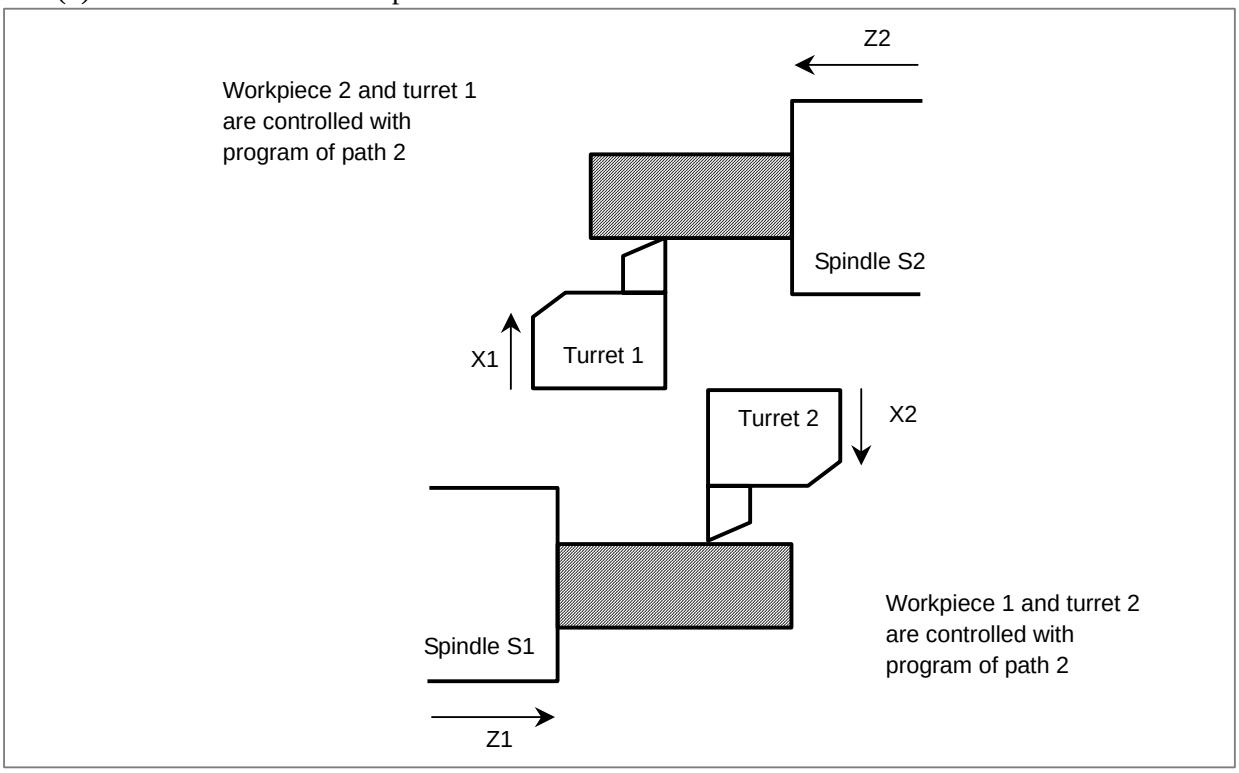
- Examples of independent control and of interpolation on X1-Z2 and X2-Z1

(1) Machine configuration

(a) Independent control

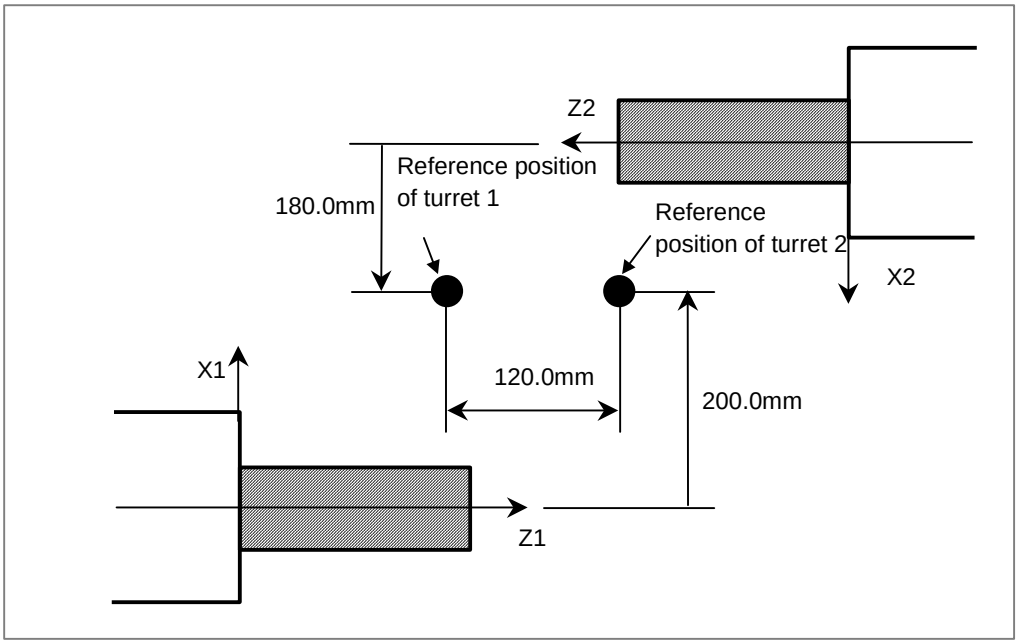


(b) X1-Z2 and X2-Z1 interpolation



(2) Parameter settings

- To perform composite control by letting the X1 and X2 axes interchange, set parameter No. 8183x of path 2 to "101".
- Because the direction of the coordinates on the X1 axis is opposite from that of the coordinates on the X2 axis, set parameter MCDx (No.8162#6) of path 2 to "1".
- To automatically set the position of the remote turret in the workpiece coordinate system of each path at the start of composite control, set parameter MPMx (No.8162#4) of each of paths 1 and 2 to "1".
- To automatically set the position of the local turret in the workpiece coordinate system of each path at the end of composite control, set parameter MPSx (No.8162#5) of each of paths 1 and 2 to "1".
- Assuming that the relations between the workpiece coordinates of each path and the reference position are as shown in the figure below, set parameter No. 8184x of path 1 to "200000" and No. 8184x of path 2 to "180000".



(3) Signal operation

- To start composite control on the X2 and X1 axes, set signal MIX1 <G1128.0> to "1".
- To cancel composite control, set signal MIX1 <G1128.0> to "0".
- When an emergency stop, NC reset, or alarm occurs, set signal MIX1 <G1128.0> to "0".
- Set signals other than signal MIX1 <G1128.0> to "0".

(4) Program example

```
Path 1                Path 2
N1000  . . .          N2000  . . .        Independent machining of each path
N1010  M200 ;         N2010  M200 ;       Waiting
                      N2020  M55 ;        Start composite control on the X1 and X2 axes
N1030  M201 ;         N2030  M201 ;       Waiting
N1040  T0313  ;       N2040  T0212  ;     Selection of composite control tool and setting of offset
N1050  G50  W120. ;   N2050 G50 W120. ;   Shifting of Z axis workpiece coordinate system
N1060  S1000  M4  ;   N2060  S1500  M4  ;
N1070  G0  X20.  Z15. ;  N2070  G0  X15. Z30.;
N1080  G1  F0.5 W- 8. ;  N2080 G1 F0.1 W-5. ;     Start of composite control
N1090  . . .          N2090  . . .
N1100  M200 ;         N2100  M200 ;       Waiting
                      N2110  M56 ;        End of composite control
N1120  M201 ;         N2120  M201 ;       Waiting
N1130  G50  W- 120. ;  N2130 G50 W-120. ;  Shifting of Z axis workpiece coordinate
system
N1140  . . .          N2140  . . .        Independent machining of each path
```

where

M55 is an M code to start composite control (set signal MIX1 <G1128.0> to "1"), and
M56 is an M code to cancel composite control (set signal MIX1 <G1128.0> to "0").

NOTE

At the start and end of composite control, you do not necessarily perform automatic setup of the coordinate system. If you do not perform automatic setting, the program will set an appropriate one internally.

Others

- In the preceding explanation, a single synchronous control or composite control operation is explained. In reality, however, you can select multiple synchronous control or composite control operations and execute them at the same time: Set all necessary parameters in advance, and specify which synchronous or composite control to perform, using appropriate signals. Make sure that a single axis is not subject to multiple synchronous/composite control operations at the same time.
- You can set only a single set of axes subject to synchronous or composite control. If you require multiple sets, use the programmable parameter input function (G10) to change the parameter settings in the program. Before changing parameters, make sure that the associated synchronous or composite control has been canceled.

(Example)

Change parameter settings to synchronize the Z2 (slave) axis to the Y1 (master) axis.
(To set parameter No. 8180z of path 2 to "103", execute a program such as that shown below.)

```
N0200  . . .
N0210  G10  L50  ;              Start of parameter settings
N0220  N8180  P2  R103  ;       Set parameter No. 8180z to "103".
N0230  G11  ;                   End of parameter settings
N0240  . . .
```

Execute blocks G10 and G11 while the Y1 and Z2 axes are not subject to synchronous or composite control.

Troubleshooting

1. Unable to start synchronous or composite control. (No alarm occurs.)
   (1) The correct synchronous or composite control options have not been set up.
   → To perform synchronous or composite control, you must set up the correct options.
   (2) Signal Gn128 or Gn138 has not risen.
   → Synchronous or composite control starts at the rise of signal Gn128 or Gn138. After synchronous or composite control ends due to a reset or alarm, merely canceling the reset or alarm cause will not cause synchronous or composite control to restart. After canceling the reset or alarm cause, you must make the signal rise again.
   (3) The axis number of the axis subject to synchronous or composite control is not set in the appropriate parameter.
   → To perform synchronous control, set the axis number of the master axis in parameter No. 8180. To perform composite control, set the axis number of the interchanging axis in parameter No. 8183. If using a 2-path interface, use the parameter of path 2.
   (4) Synchronous or composite control cannot be started if the NC unit is one of the following states:
   - Emergency stop
   - Reset
   - Servo alarm
   - Alarm PW0000
   - Alarm related to synchronous or composite control

   Synchronous or composite control cannot be started, either, if the axis on which synchronous or composite control is to start is either of the following states:
   - Servo off
   - Overtravel

2. An attempt to raise a synchronous or composite start signal results in an alarm.
   (1) An attempt is made to use an axis already under synchronous or composite control for another synchronous or composite signal operation. (Alarms (PS0352, PS0357))

→ You cannot use a single axis for a combination of multiple synchronous or composite control operations. Note, however, that the synchronous master axis can be the master of multiple synchronous slave axes at the same time. You can add additional slave axes while synchronous control is in progress.

(2) The axis number set in a parameter is greater than the number of controllable axes. (Alarms (PS0350, PS0355))

→ The number of the synchronous master axis and that of the interchanging axis subject to composite control must not exceed the number of axes controllable in the remote path (or the local path if you are to perform synchronous control in that path).

(3) The tool is moving along the axis that you want to control. (Alarms (PS0351, PS0356))

→ At the start of synchronous or composite control, the tool must be stopped on the axis subject to that control. That the tool is stopped on an axis means that the speed specified after acceleration/deceleration is zero.

3. An attempt to cancel synchronous or composite control results in alarms (PS0351, PS0356).

(1) The tool is moving along the axis on which the control you want to cancel.

→ To cancel synchronous or composite control, the tool must be stopped on the axis on which to cancel it. That the tool is stopped on an axis means that the speed specified after acceleration/deceleration is zero. Before canceling, make sure that the movement in progress signal Fn102 has changed to "0".

4. During synchronous control or composite control, an alarm occurs.

(1) A move command is issued for a synchronous slave axis. (Alarm (PS0351))

→ No move command can be issued for a synchronous slave axis, whether it be automatic or manual.

(2) A move command is issued for an axis under composite control for which parameter MUMx (No. 8162#7) is set to "1". (Alarm (PS0353))

→ During composite control, no move command can be issued for an axis for which parameter MUMx (No. 8162#7) is set to "1", whether it be automatic or manual.

(3) During synchronous control, an automatic reference position return command with G28 is issued for a parking master axis. (Alarm (PS0354))

→ In this case, a reference position return is performed on the slave axis, requiring that the reference position of the slave axis be established.

(4) During composite control, an automatic reference position return command with G28 is issued. (Alarm (PS0359))

→ In this case, a reference position return is performed on the remote interchanging axis, requiring that the reference position of the interchanging axis be established.

5. During synchronous control, servo alarm (SV0407) occurs.

(1) The difference between the synchronous master and slave axes in positional deviation is too large.

→ Servo alarm (SV0407) will occur in a synchronous error check. In general, this alarm occurs if the difference between the synchronous master and slave axes in acceleration/deceleration time constant or servo parameter is large. It can also occur if the actual movement of the machine is not correct (is not correctly synchronized) due to some cause or other.

6. During synchronous control, the machine position deviates.

(1) The synchronous master and slave axes in acceleration/deceleration time constant or servo parameter (such as loop gain) differs.

→ In this synchronous control, acceleration/deceleration control and servo control are performed in the master and slave independently of each other. (That is, move commands are synchronized.) For this reason, if the master and slave differ in acceleration/ deceleration time constant and servo characteristics, the actual movement of the machine may not fully match.

7. The travel distance is not correct during synchronous or composite control.
   (1) The master and slave axes differ in diameter/radius specification or inch/millimeter setting.
   → In synchronous or composite control, diameter/radius conversion or inch/millimeter conversion is not performed between master and slave. The master and slave must match in least command increment.

8. After a switch to synchronous or composite control, the tool does not move to the specified coordinates.
   (1) After a switch to synchronous or composite control, a move command is issued without waiting (with an M code without buffering).
   → If, during automatic operation, you want to start or cancel synchronous or composite control with automatic coordinate system setting, be sure to perform waiting (with an M code without buffering) before and after that.

9. Synchronous or composite control is canceled even when <Gn128> or <Gn138> has not been raised.
   → In synchronous or composite control, synchronous or composite control will be automatically canceled if at least either of the paths enters one of the following states:
   - Emergency stop
   - Reset
   - Servo alarm
   - Alarm (PW0000)
   - Alarm related to synchronous or composite control

   In addition, if at least one of the axes on which synchronous or composite control is in progress enters either of the following states:
   - Servo off
   - Overtravel

10. Alarm (DS1933) occurs.
   → In synchronous or composite control, alarm (DS1933) occurs if at least one path enters one of the following states:
   - Emergency stop
   - Servo off
   - Servo alarm

   After canceling the emergency stop, after canceling servo off, or after restoring from the servo alarm, first perform a reference position return or coordinate system setting and then start synchronous or composite control.

8.5.3 Hypothetical Cs Axis Control

Overview

This function allows you to add a hypothetical Cs axis to a path to which no serial spindle is actually connected.
Usually, performing composite control on Cs axes in 2-path control requires that both of the axes to be interchanged between paths be Cs axes. Using this function, however, enables composite control in which a hypothetical Cs axis and a Cs axis to which a serial spindle is actually connected interchange with each other. This makes it possible to move the serial spindle with Cs contour control that is connected to another path with a command to move from a path to which a serial spindle is not connected to a hypothetical Cs axis.

Explanation

By setting parameter CDMx (No. 1014#7) to 1 and setting a Cs axis in parameter No. 1023, you can add the hypothetical Cs axis to a path to which a serial spindle is not actually connected.

In 2-path control, this enables composite control in which a hypothetical Cs axis and an actual Cs axis connected to another path interchange with each other.

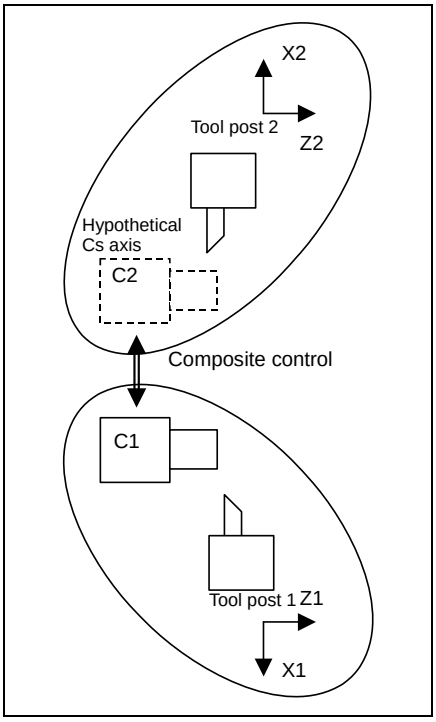
For the hypothetical Cs axis, you need not set up a Cs contour control mode switching signal.

NOTE
1 An attempt to turn ON the Cs contour control mode switching signal for a hypothetical Cs axis results in alarm SP0752.
2 If a program issues a move command for a hypothetical Cs axis for which composite control is OFF or for a hypothetical Cs axis and a Cs axis under composite control, alarm PS0197 will occur.

Example of control in which a hypothetical Cs axis and composite control are combined

1) Control in which a Cs contour control axis (a single serial spindle) is used in two path

[Machining pattern]
Usually, machining is performed in path 1, using X1-Z1-C1.
By performing composite control on C1-C2, you can operate C1 (Cs axis) with a C2 command in path 2, thereby enabling machining using X2-Z2-C1.



Axis configuration

Path 1	Path 2
X1 (servo axis)	X2 (servo axis)
Z1 (servo axis)	Z2 (servo axis)
C1 (Cs-axis)	C2 (hypothetical Cs axis)

Parameter setting

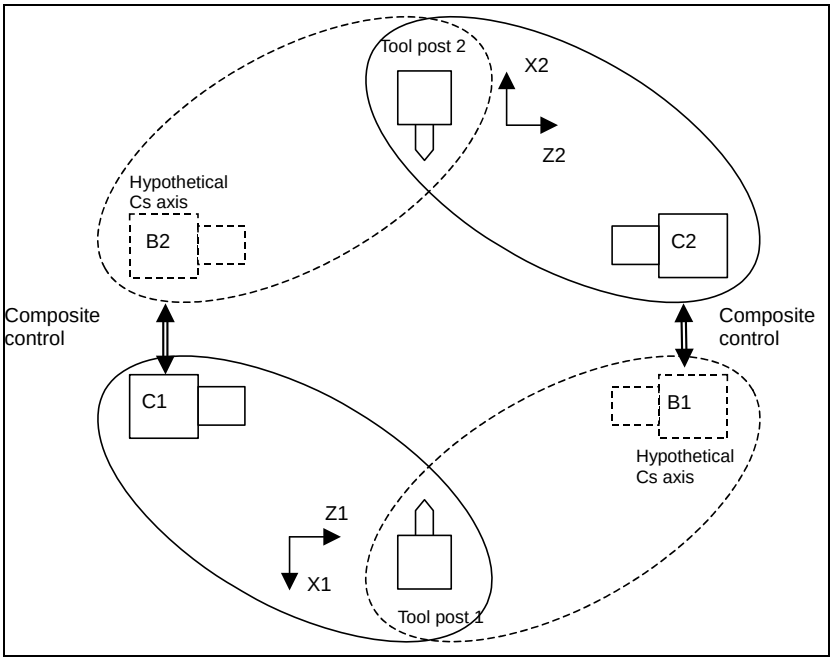
		Parameter No. 1023 (servo axis number)	Parameter No. 1014#7 (Hypothetical Cs axis setting)
1st axis	Path 1, axis 1, X1	1	0
2nd axis	Path 1, axis 2, Z1	2	0
3rd axis	Path 1, axis 3, C1	-1	0

		Parameter No. 1023 (servo axis number)	Parameter No. 1014#7 (Hypothetical Cs axis setting)
4th axis	Path 2, axis 1, X2	3	0
5th axis	Path 2, axis 2, Z2	4	0
6th axis	Path 2, axis 3, C2	-2	1

		Parameter No. 3717 (spindle motor number)
1st spindle	Path 1, spindle 1	1
2nd spindle	Path 2, spindle 1	0

2) Control in which Cs contour control axes (two serial spindle) are used in two paths

[Machining pattern]
Usually, machining is performed in path 1, using X1-Z1-C1, and in path 2, using X2-Z2-C2.
By performing composite control on B1-C2, you can operate C2 (Cs axis) with a B1 command in path 1, thereby enabling machining using X1-Z1-C2.
By performing composite control on C1-B2, you can operate C1 (Cs axis) with a B2 command in path 2, thereby enabling machining using X2-Z2-C1.



Axis configuration

Path 1	Path 2
X1 (servo axis)	X2 (servo axis)
Z1 (servo axis)	Z2 (servo axis)
C1 (Cs-axis)	C2 (Cs-axis)
B1 (hypothetical Cs axis)	B2 (hypothetical Cs axis)

Configuration in which by using a hypothetical Cs axis, the addresses of commands for the Cs axis in the remote path are set separately from those of commands for the Cs axis in the local path.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1014	CDMx							

[Input type] Parameter input

[Data type] Bit axis

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#7 CDMx The Cs contour control axis is:
0: Not a hypothetical Cs axis
1: Hypothetical Cs axis

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 Number of controlled axes
Set the servo axis for each control axis.
Usually set to same number as the control axis number.
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 control axis by using the first spindle, set -1.

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.
3: Spindle motor connected to amplifier number 3 is used.

Alarm and message

Number	Message	Description
PS0197	C-AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cs-axis when the Cs contour control switching signal was off.
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.

Caution

⚠ CAUTION

1 Set the necessary parameters such as feedrate for the hypothetical Cs axis as well. You need not set serial spindle parameters (such as Nos. 4000 to 4799).
2 For Cs contour control, see the explanation of each function.

Note

NOTE

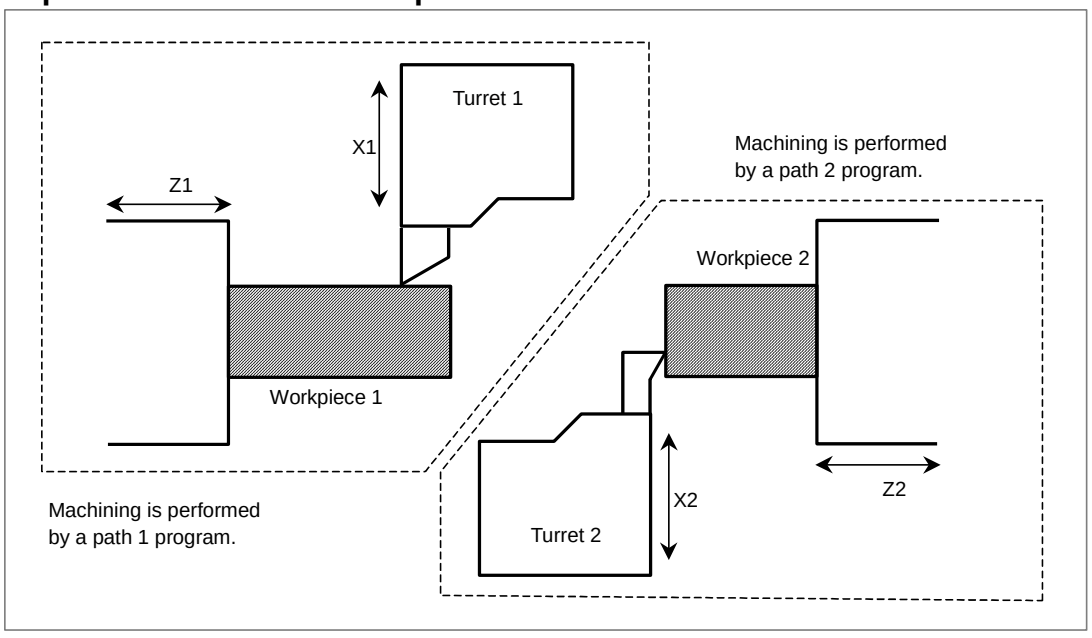
To use this function requires the synchronous/composite control option.
In a number of controlled axes specification and a number of controlled spindles specification, you must include the hypothetical Cs axis. (The maximum number of controllable axes and the maximum number of controllable spindles in the system or in each path must not be exceeded.)

8.6 SUPERIMPOSED CONTROL

Overview

The superimposed control function adds the amount of movement of an axis (superimposed control master axis) in one path to an axis (superimposed control slave axis) on the other path for which ordinary move commands are being executed. This function is similar to synchronous control but differs from it in that move commands can be issued not only for the master axis but also for the slave axis. The slave axis moves by the sum of the amount of movement specified by its own move commands and the amount of movement specified by move commands for the master axis. Appropriate setting of parameter OMRx (No. 8162#3) can reverse the direction in which the master and slave axes move.

- Independent control of each path



This function enables superimposed control between paths and in a single path, as described below.

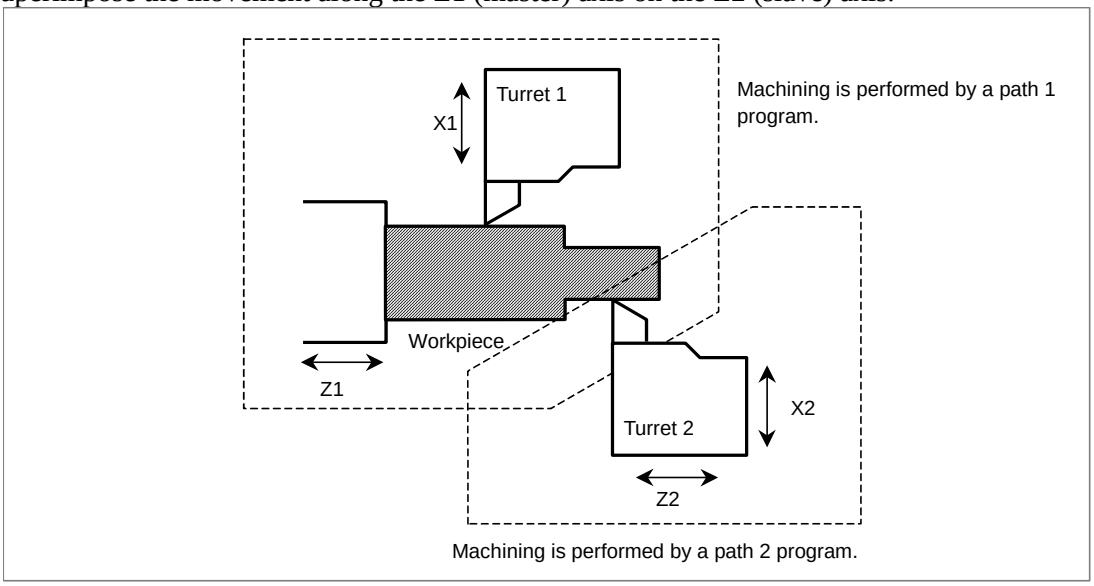
NOTE

Superimposed control is optional function.

- Superimposed control

The move command for any axis is superimposed on an axis of another path.
(Example)

Superimpose the movement along the Z1 (master) axis on the Z2 (slave) axis.



Explanation

- Setting

Specify which axis is to be the master axis to be under superimposed control, with parameter No. 8186, using the path number and the axis number.
Example)

For an axis configuration in which all paths consist of X, Z, and Y axes
To superimpose the movement of the Z1-axis (master) to that of the Z2-axis (slave) :
Parameter No. 8186z of path 2 = 102
To superimpose the movement of the X1-axis (master) to that of the Z2-axis (slave) :
Parameter No. 8186z of path 2 = 101
To superimpose the movement of the X1-axis (master) to that of the Y1-axis (slave) :
Parameter No. 8186y of path 1 = 101

- Programming

Both before and after the M codes for a start and a cancellation of superimposed control, specify a waiting M code.

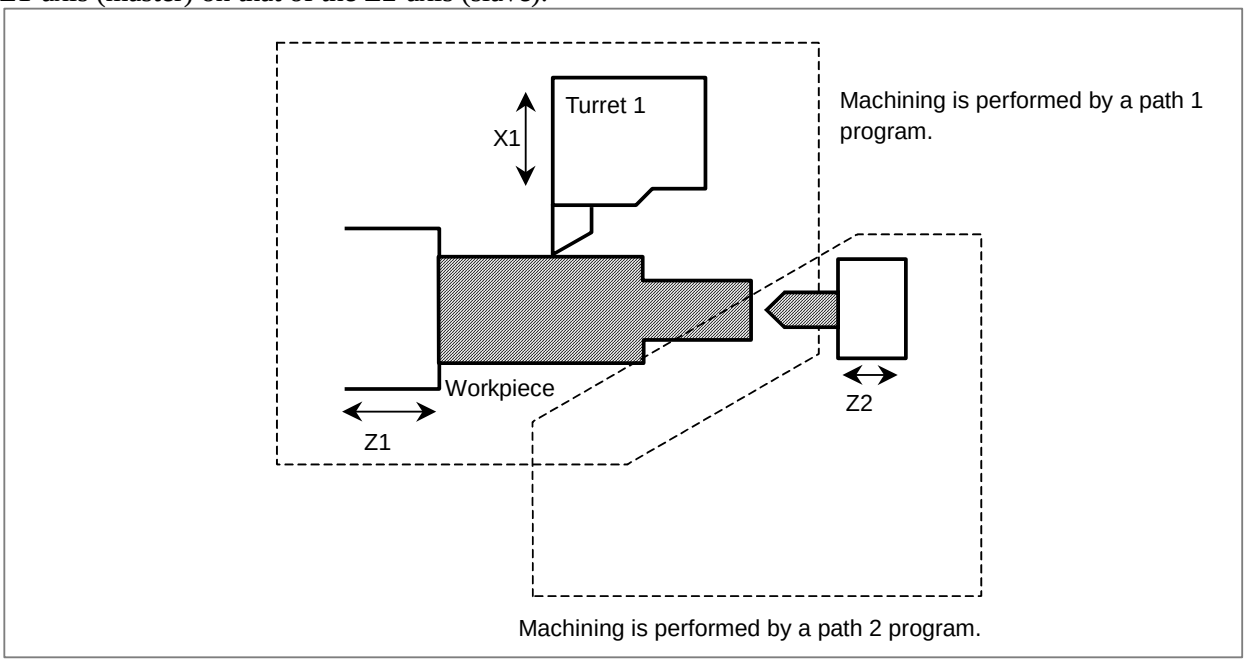
Master axis path	Slave axis path	
:	:	
M100;	M100;	Waiting
	M220;	Start of superimposed control
M101;	M101;	Waiting
:	:	
Independent operation	Independent operation	
:	:	
M100;	M100;	Waiting
	M221;	Cancellation of superimposed control
M101;	M101;	Waiting
:	:	

- Signal operation

When superimposed control begins or ends (when an M code is issued), the superimposed control axis selection signals OVLS1 to OVLS5 for the target slave axis (from the PMC to the CNC) are changed from "0" to "1" (to begin superimposed control) or from "1" to "0" (to terminate superimposed control).

- Examples of applications

Suppose that a workpiece on the spindle (Z1-axis) that moves along the axis is to be cut with a tool in path 1 and a tool in path 2 simultaneously. This example superimposes the amount of movement of the Z1-axis (master) on that of the Z2-axis (slave).



- Feedrate

Because the amount of movement of the master axis is added to that of the slave axis, the resulting speed of the slave axis may become much larger than a normal speed. ("Normal speed" refers to such a feedrate as a rapid traverse rate set by a parameter.)

- Differences between superimposed control and ordinary synchronous control

- Neither out-of-synchronization compensation or detection is performed between the master and slave axes during superimposed control.
- A parking signal is ineffective for axes under superimposed control.
- A reference position return cannot be specified for the salve axis under superimposed control.

- Follow-up

If an emergency stop or servo alarm occurs during superimposed control, the follow-up operation is performed on machine coordinates. After the emergency stop or servo alarm is canceled, the workpiece coordinate system is preset.

To enable this function, set the following parameters:

- Set bit 7 (NRS) of parameter No. 8160 to 0.
- Set bit 1 (MSO) of parameter No. 8168 to 1.
- Set bit 5 (OPSx) of parameter No. 8164 to 1.

NOTE

1 Servo off generates alarm DS1933 "NEED REF RETURN(SYNC:MIX:OVL)". After canceling the alarm, perform the manual return to the reference position.

NOTE
2 To preset workpiece coordinates, use a workpiece coordinate system (set 0 in bit 0 (NWZ) of parameter No .8136). If there is no workpiece coordinate system (1 is set in bit 0 (NWZ) of parameter No .8136), manual return to the reference position is required. Even if 1 is set in bit 5 (OPSx) of parameter No. 8164, absolute coordinates cannot be restored normally after the emergency stop or servo alarm is canceled.

Signal

Superimposed control axis selection signals OVLS1 to OVLS5 <Gn190.0 to Gn190.4>

[Classification] Input signal
[Function] These signals perform superimposed control.
[Operation] When one of these signals becomes "1", the control unit:
- Begins superimposed control over the corresponding axis.
  The master axis is selected according to parameter No. 8186.

Synchronous/composite/superimposed control under way signals SYN1O to SYN5O <Fn118.0 to Fn118.4>

[Classification] Output signal
[Function] These signals indicate each axis is being subjected to synchronous/composite/superimposed control.
[Output cond.] These signals become "1" under the following condition:
- The corresponding axis is under synchronous, composite, or superimposed control.

These signals become "0" under the following condition:
- The corresponding axis is not under synchronous, composite, or superimposed control.

⚠ CAUTION
Whether each axis is under superimposed control does not always match whether the superimposed control axis selection signal has been issued or not. For example, if these signals are set to "1" during an alarm, they are ignored. If a servo alarm occurs during superimposed control, they are terminated automatically. Before attempting to perform superimposed control, always check the state of these signals.

Superimposed control master axis confirmation signals OVMO1 to OVMO5 <Fn344.0 to Fn344.4>

[Classification] Output signal
[Function] These signals notify whether the corresponding axes are superimposed control master axes.
[Output cond.] These signals become "1" under the following condition:
- The corresponding axes are superimposed control master axes.

These signals become "0" under the following condition:
- The corresponding axes are released from superimposed control.

Superimposed control slave axis confirmation signals OVSO1 to OVSO5 <Fn345.0 to Fn345.4>

[Classification] Output signal
[Function] These signals notify whether the corresponding axes are superimpose control slave axes.

[Output cond.] These signals become "1" under the following condition:
- The corresponding axes are superimposed slave axes.

These signals become "0" under the following condition:
- The corresponding axes are released from superimposed control.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn190				OVLS5	OVLS4	OVLS3	OVLS2	OVLS1
	#7	#6	#5	#4	#3	#2	#1	#0
Fn118				SYN5O	SYN4O	SYN3O	SYN2O	SYN1O
Fn344				OVMO5	OVMO4	OVMO3	OVMO2	OVMO1
Fn345				OVSO5	OVSO4	OVSO3	OVSO2	OVSO1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8160	NRS		NCS	AXS				

[Input type] Parameter input
[Data type] Bit path

#4 AXS When the axis movement in-progress signal <Fn102> or the axis movement direction signal <Fn106> of the slave axis in superimposed control is output:
- 0: State output is performed according to the result of adding superimposed move pulses.
- 1: State output is performed according to the result of movement along each axis instead of superimposed move pulses.

#5 NCS If an overtravel alarm is issued for an axis under synchronous, composite, or superimposed control, synchronous, composite, or superimposed control is:
- 0: Released.
- 1: Not released.

NOTE
If one of these paths is 1, both paths are assumed to be 1.

#7 NRS When the system is reset, synchronous, composite, or superimposed control is:
- 0: Released.
- 1: Not released.

	#7	#6	#5	#4	#3	#2	#1	#0
8162					OMRx			

[Input type] Parameter input
[Data type] Bit axis

#3 OMRx Superimposed mirror-image control is:
- 0: Not applied. (The superimposed pulse is simply added.)
- 1: Applied. (The inverted superimposed pulse is added.)

	#7	#6	#5	#4	#3	#2	#1	#0
8164		SOKx	OPSx					

[Input type] Parameter input
[Data type] Bit axis

#5 OPSx When superimposed control is canceled, control in which an amount of movement along a master axis subject to superimposed control is added to the workpiece coordinate of a slave axis is:
0: Not applied.
1: Applied.

NOTE
When the workpiece coordinate system is enabled (bit 0 (NWZ) of parameter No. 8136 is 0), the coordinate system is set by workpiece coordinate system presetting (equivalent to G92.1 IP0).

#6 SOKx If a master axis subject to superimposed control is also subject to synchronous control:
0: An alarm is issued when superimposed control is started during synchronous control.
1: No alarm is issued when superimposed control is started during synchronous control.

	#7	#6	#5	#4	#3	#2	#1	#0
8168				MWR		SVF	MSO	MPA

[Input type] Parameter input
[Data type] Bit

#0 MPA When an alarm about synchronization/composite/superposition control is issued:
0: Both paths are set to the feed hold state.
1: Only the path that includes an axis related to synchronization/composite/superposition control is set to the feed hold state.
For example, in synchronization control within the same path, the path in which an alarm occurred is set to the feed hold state.

#1 MSO When one of the following events occurs in synchronous control or composite control:
- The emergency stop signal is turned off.
- The servo-off signal is turned on.
- A servo alarm is issued.

0: The synchronous or composite control mode is canceled and follow-up operation is not performed.
For the operation to be performed when the servo-off signal is turned on, however, the setting of bit 7 (NSR) of parameter No. 8161 is used in synchronous control or the setting of bit 0 (NMR) of parameter No. 8161 is used in composite control.

1: The synchronous or composite control mode is not canceled. The following operation is performed to perform follow-up operation:
When the emergency stop signal is turned off, if the other path is involved, the path operates so that the emergency stop signal is virtually turned off.
When the servo-off signal is turned on, the relevant axis is determined and operation is performed so that the servo-off signal is virtually turned on for the determined axis.
When a servo alarm is issued, the relevant axis is determined and the alarm SV0003, "CONTINUATION OF SYNCHRONOUS OR COMPOSITE CONTROL DISABLED" is issued for the determined axis to stop moving the tool along the axis.
When bit 2 (SVF) of parameter No. 8168 is set to 1, this servo-off specification follows the SVF setting.

NOTE
This setting is valid also during operation. For all axes placed under synchronous or composite control, the emergency stop signal is turned off, the servo-off signal is turned on, or a servo alarm is issued.

#2 SVF When an axis under composite control is placed in the servo-off state:
0: Composite control is canceled.
1: Composite control is not canceled.

Follow-up specification follows the setting of bit 0 (FUPx) of parameter No. 1819.
When bit 2 (SVF) of parameter No. 8168 is set to 1, bit 0 (NMR) of parameter No. 8161 is invalid. Bit 1 (MSO) of parameter No. 8168, specification for servo-off, is also invalid.

NOTE
If a composite control axis is placed in the servo-off state when stopped, set this parameter to 1.

#4 MWR When the workpiece coordinate system is automatically set if synchronization/composite/superposition control is started or ended, the tool compensation number is:
0: Canceled.
1: Not canceled.

NOTE
This parameter is valid when the workpiece coordinate system is enabled (bit 0 (NWZ) of parameter No. 8136 is 0).

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, . . .)
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 (U2 of path 2):
The travel distance of X1 is superimposed on X2, and the travel distances of X1 and X2 are further superimposed on U2.
Parameter No. 8186x of path 2 = 101
Parameter No. 8186u of path 2 = 201

Alarm and message

If any of the following alarms occurs, superimposed control will be released for all axes.

- P/S alarm

Number	Message	Description
PS0360	PARAMETER OF THE INDEX OF THE SUPERPOS CONTROL AXIS SET ERROR.	An illegal superposition control axis number (parameter No. 8186) is specified.
PS0361	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.
PS0362	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.
PS0363	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.
PS0364	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.

- D/S alarm

Number	Message	Description
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.

Caution

⚠ CAUTION

1 At the start or end of superimposed control, the tool must be stopped on the axis subject to that control.

⚠ CAUTION

2 Before or after an M code for starting or canceling superimposed control during automatic operation, be sure to specify a waiting M code (M code without buffering). When exercising superimposed control in a path, be sure to prohibit look-ahead operation by specifying an M code without buffering before or after an M code for starting or canceling superimposed control.
3 The axes under superimposed control must match in least command increment, Detection unit, and diameter/radius specification. Otherwise, the amounts of travel will not be equal.
4 In superimposed control, do not change the parameters related to it (including axis control, increment system, feedrate, and acceleration/deceleration control).
5 Before starting superimposed control, make sure that the axes subject to it has undergone a reference position return after the power was turned on or that a reference position has been established with an absolute pulse coder.
6 If an emergency stop or servo off is performed or if a servo alarm occurs, the coordinates will change due to synchronous or composite control. If this occurs, "alarm DS 933" will be generated. After canceling the emergency stop, canceling servo off, or restoring from the servo alarm, perform a reference position return and coordinate system setting first, then start superimposed control.
7 In superimposed control, pitch error compensation, backlash compensation, and stored stroke check are performed on the position resulting from adding superimposing pulses.
8 The following servo software functions cannot be used with superimposed control:
   - Advance preview feed forward function
   - Abnormal load detection and switching function for each of cutting and rapid traverse
   - Cutting/rapid velocity gain switching function
   - Electric current 1/2 and PI switching function for each of cutting and rapid traverse
   - Torque command filter switching function for each of cutting and rapid traverse
9 The following functions cannot be used in superimposed control:
   - Advanced preview control
10 During superimposed control, reference position establishment based on "linear scale with absolute address reference mark" or "linear scale with absolute address zero point" is impossible.

NOTE

1 You can place more than one axis under superimposed control at the same time. You cannot, however, place an axis subject to synchronous or composite control under superimposed control.
2 Superimposed control cannot be performed on a linear axis and a rotation axis.
3 The superimposed control option and axis synchronous control option can be specified at the same time.

Limitation

- Limitations on superimposed control

Function	In superimposed control
Acceleration/deceleration control	The move pulses of the superimposed control master axis after acceleration/deceleration are added to those of the slave axis after acceleration/deceleration.
Reference position return	Not allowed for a superimposed slave axis.
2nd, 3rd, 4th reference position return	Not allowed for a superimposed slave axis.
Reference position return check	Not allowed for a superimposed slave axis.
Machine coordinate system selection	Not allowed for a superimposed slave axis.
PMC axis control	Allowed.
Polar coordinate interpolation, Cylindrical interpolation	Allowed.
Handle interruption	Not allowed for a superimposed control axis.
Mirror image	Only master signals are effective to superimposing pulses.(1)
Machine lock	Only master signals are effective to superimposing pulses.(1)
Inter lock	Only master signals are effective to superimposing pulses.(1)
Override	Only master signals are effective to superimposing pulses.(1)
External deceleration	Only master signals are effective to superimposing pulses.(1)
Skip function	Allowed.
Automatic tool offset	Not allowed for a superimposed slave axis.
Follow-up	Not allowed in superimposed control.
Program restart	Not allowed for program containing superimposed control.
Cs contour control	Superimposed control is possible.(2)
Spindle positioning	Superimposed control is impossible.
Servo off	Superimposed control is impossible.

(1) For the move commands inherent to the master and slave, these functions are processed as usual; for superimposed pulses from the master axis, these functions are not applied to the slave.
(2) Limited to combinations of Cs axes. Switch to Cs mode and perform a reference position return first, then start superimposed control.

- Reading of coordinates in superimposed control

In superimposed control, the reading of custom macro system variable positional information or of current coordinates from the PMC window is as follows:

Positional information type	In superimposed control
Absolute coordinate	Reading possible. (1)
Machine coordinate	Reading possible.
End point of each block	Reading possible. (1)
Skip signal position	Reading possible. (1)

(1) Superimposing pulses are not added.

- Canceling superimposed control

Superimposed control is canceled in the event of the following, as well as when the synchronous/composite/superimposed control axis selection signal becomes "0".
(1) Emergency stop
(2) Reset
(3) Servo alarm
(4) Servo off
(5) Overtravel
(6) Alarm related to superimposed control
(7) Alarm PW0000

If one of the above events occurs in either path, all paths will be released from superimposed control. If one of the above events occurs in either path in superimposed control, the other path will automatically be placed in feedhold state (during automatic operation) or in interlock state (during manual operation).

- Axis state output signals in superimposed control

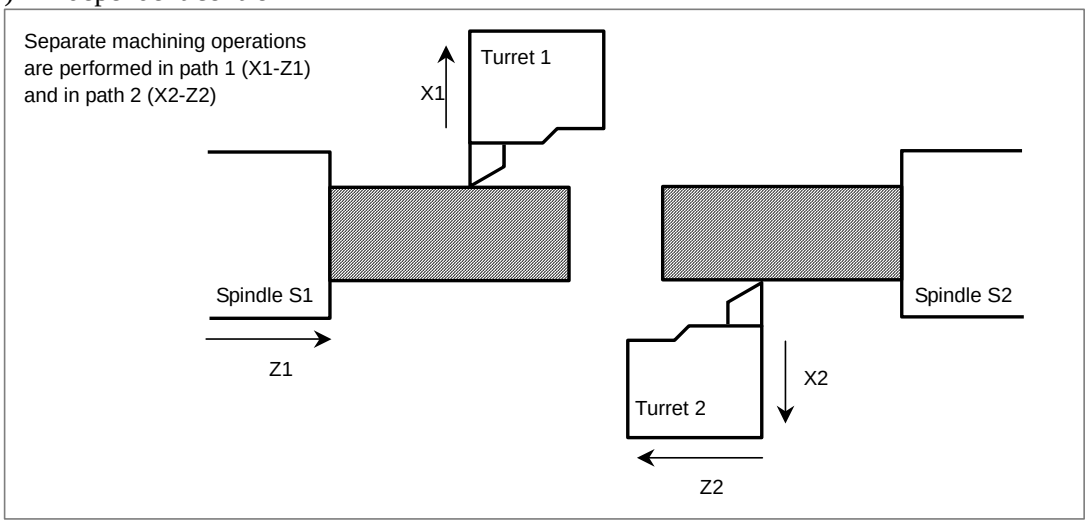
State output signal	In superimposed control
Axis moving signals MVn <Fn102>	- On the master axis, as usual. - The slave axis reflects the travel state with the command on the slave, regardless of superimposed control pulses (1)(2).
Axis moving direction signals MVDn <Fn106>	- On the master axis, the moving direction of the master axis. - For the slave axis, the moving direction after superimposed control pulses are added (2).
In-position signals INPn <Fn104>	- On the master axis, as usual. - The slave axis signal is always "1".
Reference position establishment signal ZRFn <Fn120>	- The signal on the axis for which the reference position has been established turns to "1".
Reference position return completion signal ZPn <Fn094>	- The signal value for each axis depends on the state of the axis.

(1) In a positional deviation check, regardless of the states of these signals, parameter No. 1828 is used if move command pulses are sent to the motor (regardless of whether the axis is the master or slave) as the limit, and parameter No. 1829 if none are sent.
(2) Using parameter AXS (No. 8160#4), you can switch the slave axis state output between results with superimposed control pulses added and results of individual axial movements.

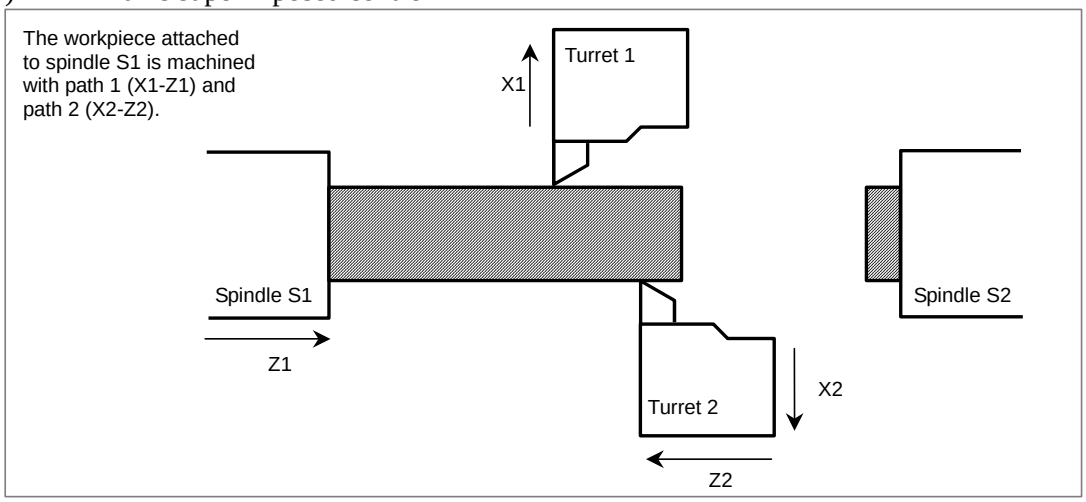
Examples of Use

- Examples of independent control and of superimposed control on the Z1 and Z2 axes

(1) Machine configuration
(a) Independent control



(b) Z1-Z2 axis superimposed control



(2) Parameter setting

- To superimpose the move command for the Z1 (master) axis on that of the Z2 (slave) axis, set parameter No. 8186z of path 2 to "102".
- Because the plus directions of the Z1 (master) and Z2 (slave) axes are opposite from each other, perform superimposed control with a mirror image applied. To do this, set parameter OMRz (No. 8162#3) of path 2 to "1".

(3) Signal operation

- To start superimposed movement along the Z1 (master) axis and the Z2 (slave) axis, set signal OVLS2 <G1190.1> to "1".
- To cancel superimpose, set signal OVLS2 <G1190.1> to "0".
- When an emergency stop, NC reset, or alarm occurs, set signal OVLS2 <G1190.1> to "0".
- Set signals other than signal OVLS2 <G1190.1> to "0".

(4) Program example

Path 1		Path 2		
N1000	. . .	N2000	. . .	Independent machining of each path
N1010	M200 ;	N2010	M200 ;	Waiting
	N2020	M55 ;		Start of superimposed control from the Z1 axis to the Z2 axis
N1030	M201 ;	N2030	M201 ;	Waiting
		N2040	T0414 ;	Selection of a superimposed control tool and setting of an offset
N1050	S1000 M3 ;			
N1060	G0 X20. Z15. ;	N2060	G0 X18. Z120. ;	Machining by turret 1 and turret 2
N1070	G1 F0.5 W-8. ;	N2070	G1 F0.1 W5. ;	
N1080	. . .	N2080	. . .	
N1010	M200 ;	N2010	M200 ;	Waiting
		N2100	M56 ;	End of superimposed control
N1010	M201 ;	N2010	M201 ;	Waiting
N1120	. . .	N2120	. . .	Independent machining of each path

where

M55 is an M code to start superimposed control, and
M56 is an M code to cancel superimposed control.

(5) Caution

⚠ CAUTION
If performing constant surface speed control, you must determine the spindle command in which path is currently effective to spindle S1.

NOTE
Input the speed of spindle S1 (feedback pulses from the position coder) to both paths 1 and 2.

Troubleshooting

1. Unable to start superimposed control. (No alarm occurs.)
   (1) The correct superimposed control options have not been set up.
   -> To perform superimposed control, you must set up the correct options.
   (2) Signal <Gn190> has not risen.
   -> Superimposed control starts at the rise of signal <Gn190>. After superimposed control ends due to a reset or alarm, merely canceling the reset or alarm cause will not cause superimposed control to restart. After canceling the reset or alarm cause, you must make the signal rise again.
   (3) The axis number of the axis subject to superimposed control is not set in the appropriate parameter.
   -> To perform superimposed control, set the axis number of the master axis in parameter No. 8186.
   (4) Superimposed control cannot be started if the NC unit is one of the following states:
   - Emergency stop
   - Reset
   - Servo alarm
   - Alarm PW0000
   - Alarm related to superimposed control

   Superimposed control cannot be started, either, if the axis on which superimposed control is to start is either of the following states:
   - Servo off
   - Overtravel
2. An attempt to raise a superimposed start signal results in an alarm.
   (1) An attempt is made to use an axis already under superimposed control for another superimposed signal operation. (Alarm PS0360)
   → You cannot use a single axis for a combination of multiple superimposed control operations.
   (2) The axis number set in a parameter is greater than the number of controllable axes. (Alarm PS0360)
   → The number of the superimposed master axis and that of the interchanging axis subject to superimposed control must not exceed the number of axes controllable in the remote path (or the local path if you are to perform superimposed control in that path).
   (3) The tool is moving along the axis that you want to control. (Alarm PS0361)
   → At the start of superimposed control, the tool must be stopped on the axis subject to that control. That the tool is stopped on an axis means that the speed specified after acceleration/deceleration is zero.
3. An attempt to cancel superimposed control results in alarms (PS0361).
   (1) The tool is moving along the axis on which the control you want to cancel.
   → To cancel superimposed control , the tool must be stopped on the axis on which to cancel it. That the tool is stopped on an axis means that the speed specified after acceleration/deceleration is zero. Before canceling, make sure that the movement in progress signal <Fn102> has changed to "0".

4. In superimposed control, the amount of travel is not correct.
   (1) The master and slave axes differ in diameter/radius specification or in inch/millimeter setting.
      → In superimposed control, diameter/radius conversion or inch/millimeter conversion is not performed between master and slave. The master and slave must match in least command increment.
5. In superimposed control, alarm (PS0363) occurs.
   (1) In superimposed control, a reference position return is performed on the slave axis.
      → No reference position return can be performed on the slave axis under superimposed control.
6. Superimposed control is canceled even when <Gn190> has not been raised.
   → In superimposed control, superimposed control will be automatically canceled if at least either of the paths enters one of the following states:
   - Emergency stop
   - Reset
   - Servo alarm
   - Alarm PW0000
   - Alarm related to superimposed control

   In addition, if at least one of the axes on which superimposed control is in progress enters either of the following states:
   - Servo off
   - Overtravel
7. Alarm DS1933 occurs.
   → In superimposed control, alarm DS1933 occurs if at least one path enters one of the following states:
   - Emergency stop
   - Servo off
   - Servo alarm

   After canceling the emergency stop, after canceling servo off, or after restoring from the servo alarm, first perform a reference position return or coordinate system setting and then start superimposed control.

8.7 SYNCHRONOUS, COMPOSITE, AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND

Overview

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 Section 8.5, "SYNCHRONOUS CONTROL AND COMPOSITE CONTROL", and Section 8.6, "SUPERIMPOSED CONTROL".

NOTE
Synchronous, composite, and superimposed control by program command is optional function.

Format

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 state 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_ ; 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

Identification numbers are unique values that set into parameter No.12600 in order to identify each axes.

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 exercised by the G51.4/G50.4 commands instead of the synchronous control axis selection signals (SYNC1 to SYNC5<Gn138.0 to 4>, n=0 to 1).

- Composite control

Composite control is exercised by the G51.5/G50.5 commands instead of the composite control axis change selection signals (MIX1 to MIX5<Gn128.0 to 4>, n=0 to 1).

- Superimposed control

Superimposed control is exercised by the G51.6/G50.6 commands instead of the superimposed control axis selection signals (OVLS1 to OVLS5<Gn190.0 to 4>, n=0 to 1).

Parameter

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, PS alarm (PS5339) occurs at G50.4/G50.5/G50.6/G51.4/G51.5/G51.6 block.

Alarm and message

Number	Message	Description
PS5339	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.

Note

NOTE

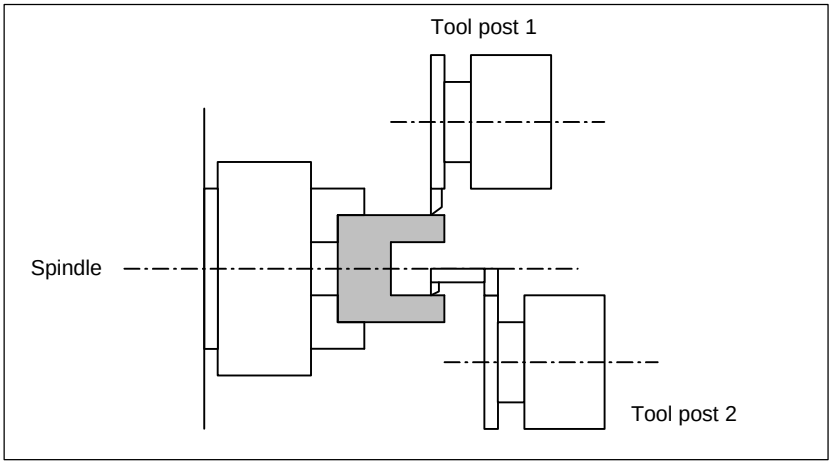
1 If a G code (G50.4/G50.5/G50.6) for canceling synchronous, composite, or superimposed control based on a program command is specified using a DI signal for an axis under synchronous, composite, or superimposed control, synchronous, composite, or superimposed control is canceled.

2 If the synchronous control axis selection signal/composite control axis change selection signal/superimposed control axis selection signal is set to 0 from 1 for an axis under synchronous, composite, or superimposed control based on a program command, synchronous, composite, or superimposed control is canceled.

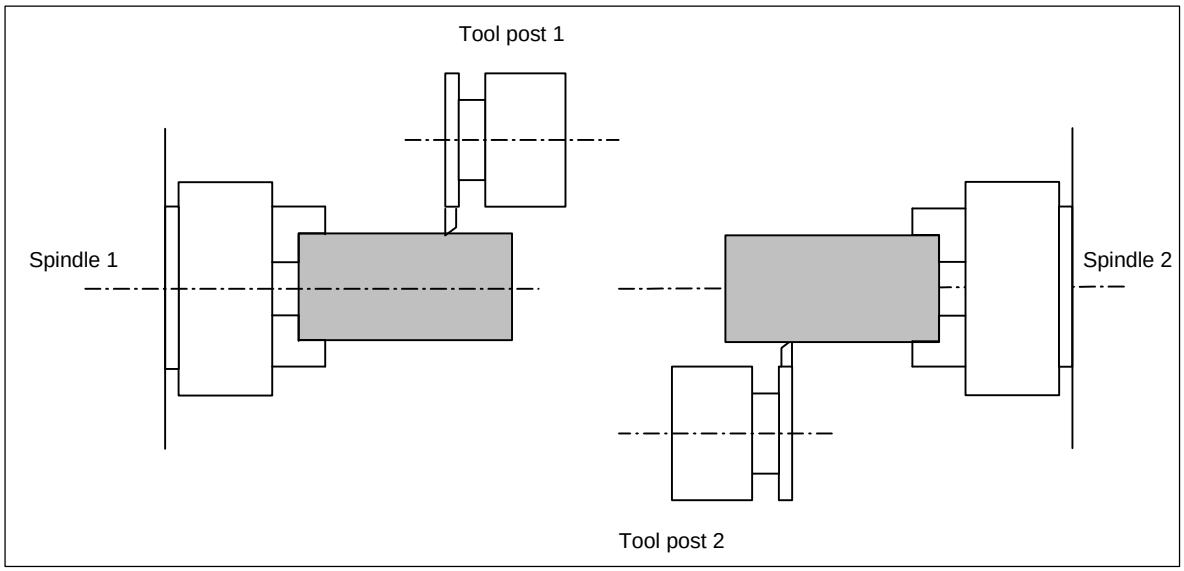
8.8 PATH SPINDLE CONTROL

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.



Application to a lathe with one spindle and two tool posts



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.

When feed per revolution is performed on each path, the feedback pulses from the position coder attached on the spindle belonging to the path are generally used. With path spindle feedback selection signals, feed per revolution can be performed using the feedback pulses from the position coder attached on the spindle belonging to any path.

NOTE
1 The spindle speed is changed by spindle commands from multiple paths because path spindle command selection signals switch to spindle commands from relevant paths. Create a PMC sequence which determines whether spindle commands programmed for each path can be executed and switches over to another path spindle command selection signal.
2 When multiple paths share one analog spindle, during execution of a thread cutting command for a path, a function (such as thread cutting or feed per revolution) which uses the feedback pulses from the spindle on another path cannot be executed normally. Put restrictions so that during execution of thread cutting on a path, feed per revolution and other functions are not executed for the tool post on another path.

Explanation

- Selecting spindle commands
  Each path spindle command selection signal (input) can be used to switch over to spindle commands for the relevant path that the spindle belonging to each path follows.

- Selecting the feedback pulses from a position coder
  When a serial spindle is used, each path spindle feedback selection signal (input) can be used to switch over to the feedback pulses from the position coder attached on the spindle belonging to the relevant path that each path uses in the NC. Thread cutting and feed per revolution using the spindle belonging to a different path can be performed.
  However, when the position coder attached to the spindle which does not exist is selected, alarm (SP1202) is issued.

The following two signal types are available as selection methods using signals and each type is specified by parameter setting as follows.

Bit 0 (2P2) of parameter No. 3703	Signal type
0	Signal type A
1	Signal type B

The using method and meaning of signals differ depending on the signal type. For details, see the explanation of signals.

Each signal type is applicable to the following spindle configurations.

Signal type	Controlling the spindle belonging to path 1 from path 1 or 2	Controlling the spindle belonging to path 1 or 2 from path 1 or 2
Signal type A	○	×
Signal type B	○	○

NOTE
When the path spindle control function is used, the spindle gear selection method must be the T type.

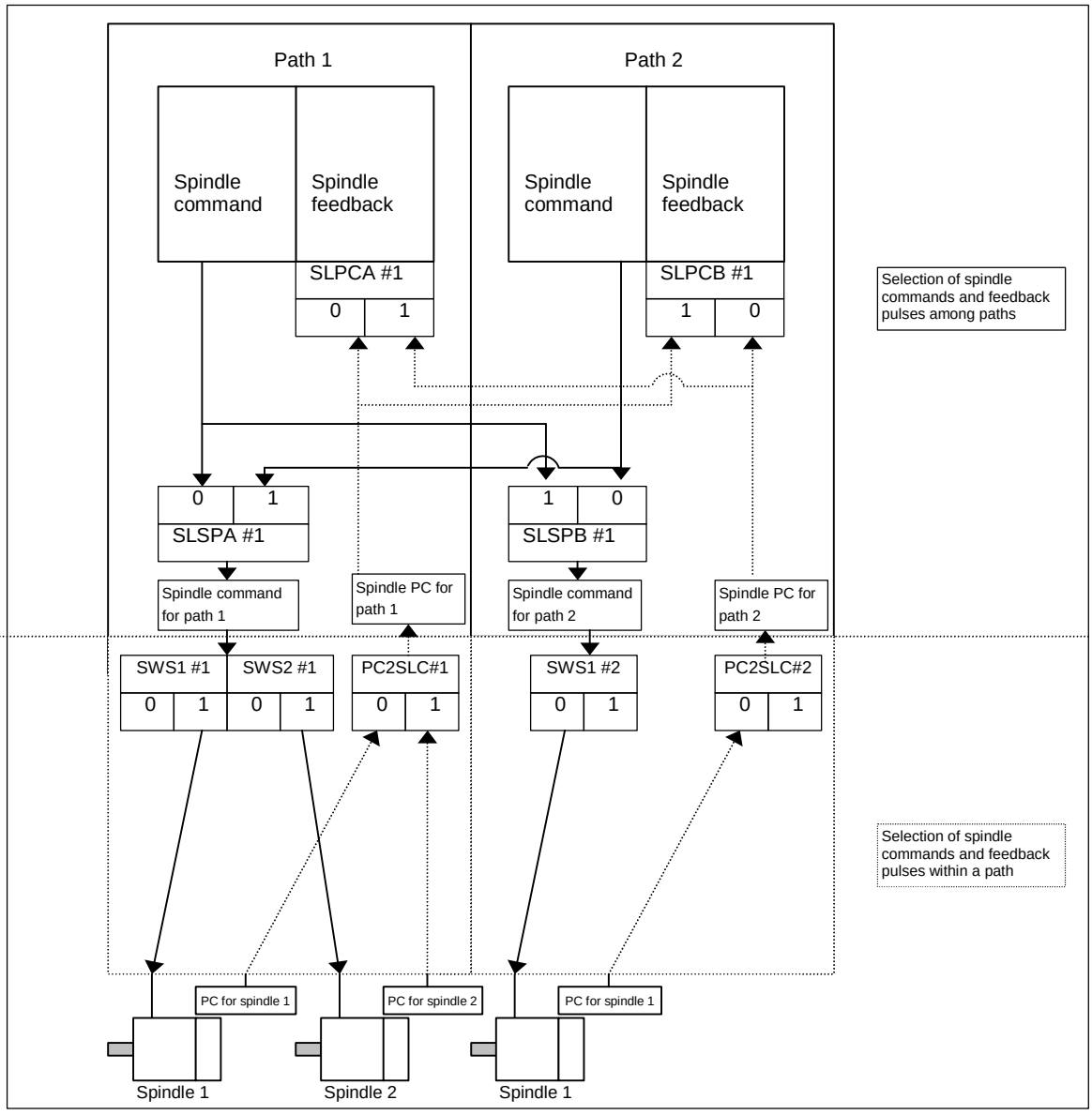
- When multiple spindles belong to individual paths

Multispindle control can be used simultaneously to issue spindle commands to any spindle belonging to individual paths.
If multispindle control is not used (0 is set in bit 3 (MSP) of parameter No. 8133), path spindle control is valid only for the first spindle belonging to each path.

The following two methods are available for selecting spindle commands in multispindle control:
1) Selecting spindle commands using signals (Bit 3 (MPP) of parameter No. 3703 is set to 0.)
2) Selecting spindle commands using address P (Bit 3 (MPP) of parameter No. 3703 is set to 1.)

When spindle commands are selected using address P, processing including path spindle command selection is performed in a 2-path system. For this reason, spindle command selection using signals is invalid (this selection is made by spindle selection signals SWS1 to SWS2 <Gn027.0 to Gn027.1> and path spindle command selection signals SLSPA <Gn063.2> and SLSPB <Gn063.3>).
Only one address P can be specified together with a spindle command and there is a one-to-one correspondence between address P values and spindles. For this reason, a spindle command in the same block cannot control multiple spindles simultaneously.

Selecting the feedback pulses from a position coder
The simultaneous use of multispindle control enables feed per revolution using the feedback pulses from the position coder on any spindle belonging to each path. Use signals (combination of position coder selection signal and path spindle feedback selection signal) to select the feedback pulses from a position coder.



Configuration in which three spindles belonging to paths 1 and 2 are shared between the two paths (Path spindle control (signal type B) + multispindle control)

Signal

There are two signal types, which are selected with appropriate parameter settings.

Parameter 2P2 (No.3703#0)	Signal type
0	Signal type A
1	Signal type B

- Signal type A

Path spindle command selection signal SLSPA <G0063.2>

[Classification] Input signal

[Function] This signal specifies the path of the program commands to be effective to the spindles belonging to path 1.

[Operation]

Signal input SLSPA<G0063.2>	Effective path of the program command
0	Spindle command of path 1
1	Spindle command of path 2

Path spindle feedback selection signal SLPCA<G0064.2>, SLPCB<G0064.3>

[Function] You need not control the path spindle feedback selection signal because the spindle feedback signal of path 1 is always effective to paths 1 and 2.

Path spindle command confirmation signal COSP <F0064.5>

[Classification] Output signal

[Function] This signal notifies which path the spindle command last specified came from.

[Output cond.] This signal becomes "1" under the following condition:

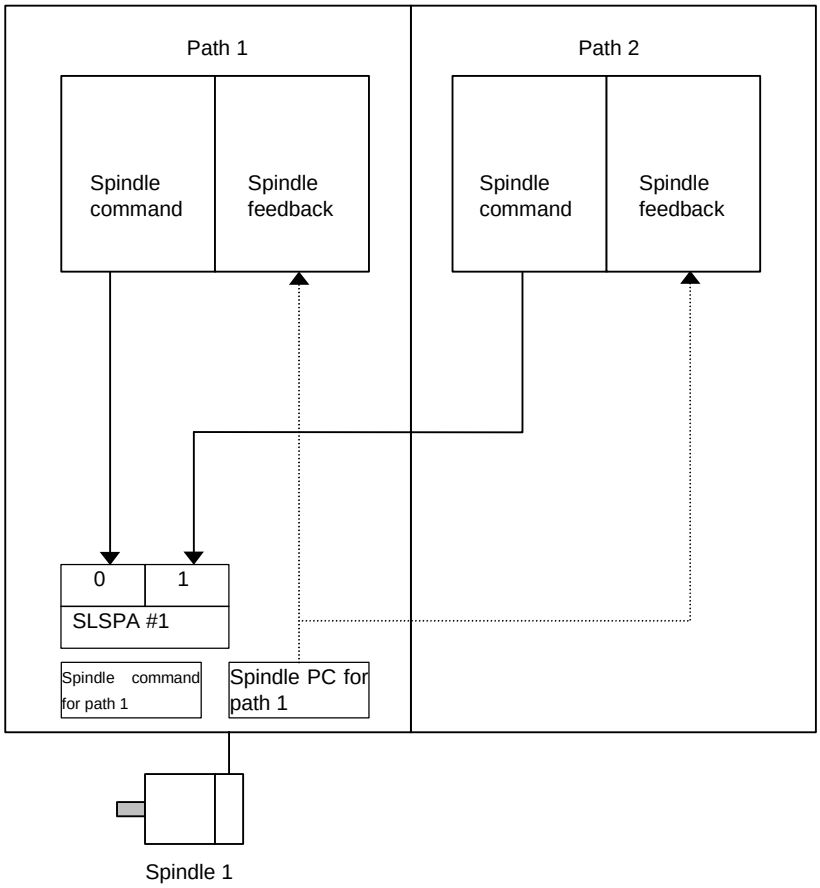
- A spindle command is issued from path 2.

This signal becomes "0" under the following condition:

- A spindle command is issued from path 1, or no spindle command is issued from either path.

In the state in which spindle commands for both paths 1 and 2 are effective to the spindles belonging to path 1, this signal allows you to determine which path the spindle command last specified came from.

NOTE
Spindle commands refer to S code commands, maximum speed command (G50S), M03/M04/M05, and commands for the constant surface speed control (G96 and G97).



Configuration in which the spindle of path 1 is shared by 2 paths

- Signal type B

Path spindle command selection signal SLSPA<G0063.2>,SLSPB<G0063.3>

[Classification] Input signal

[Function] This signal specifies the path of the program commands to be effective to the spindles belonging to paths 1 and 2.

[Operation]

Signal input SLSPA<G0063.2>	Program command to be effective to the spindle belonging to path 1
0	Spindle command of path 1
1	Spindle command of path 2

Signal input SLSPB<G0063.3>	Program command to be effective to the spindle belonging to path 2
0	Spindle command of path 2
1	Spindle command of path 1

Path spindle feedback selection signal SLPCA<G0064.2>,SLPCB<G0064.3>

[Classification] Input signal

[Function] In paths 1 and 2, this signal specifies the path to which the spindle to which position coder feedback pulses are to be effective belongs.

[Operation]

Signal input SLPCA<G0064.2>	In path 1, the path to which the spindle to which position coder feedback pulses are to be effective belongs
0	Spindle PC of path 1
1	Spindle PC of path 2

Signal input SLPCB<G0064.3>	In path 2, the path to which the spindle to which position coder feedback pulses are to be effective belongs
0	Spindle PC of path 2
1	Spindle PC of path 1

Path spindle command confirmation signal COSP<F0064.5>

[Classification] Output signal

[Function] This signal notifies which path the spindle command last specified came from.

[Output cond.] This signal becomes "1" under the following condition:

- A spindle command is issued from path 2.

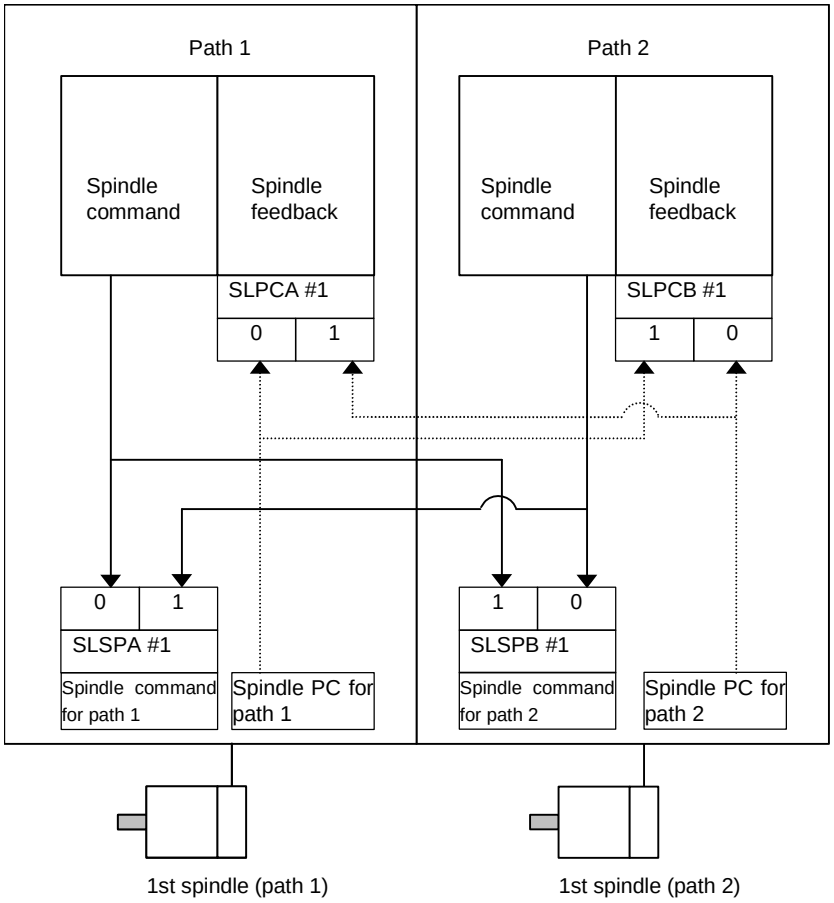
This signal becomes "0" under the following condition:

- A spindle command is issued from path 1, or no spindle command is issued from either path.

In the state in which spindle commands for both paths 1 and 2 are effective to the spindles belonging to path 1, this signal allows you to determine which path the spindle command last specified came from.

NOTE

Spindle commands refer to S code commands, maximum speed command (G50S), M03/M04/M05, and commands for the constant surface speed control (G96 and G97).



Configuration in which the spindles in paths 1 and 2 are shared by 2 paths

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0063					SLSPB	SLSPA		
G0064					SLPCB	SLPCA		

	#7	#6	#5	#4	#3	#2	#1	#0
F0064			COSP					

- Spindle command path specification signal

Spindle command path specification signal SPSP<Gn536.7>

[Classification] Input signal

[Function] This signal turns on or off the path specification of the spindle command.

[Operation] When address P is specified in multi-spindle control (1 is set in bit 3 (MPP) of parameter No. 3703)

- When this signal is set to "1", the path specified in the spindle command becomes valid.
  The spindle command can be executed from the path set in parameter No. 11090.
- When this signal is set to "0", the path specified in the spindle command becomes invalid.
  The spindle command can be executed from each path regardless of the setting of parameter No. 11090.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn536	SPSP							

Parameter

	#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.

NOTE
1 Set this parameter for a path that does not require multi-spindle control in a 2-path control system.
2 To use this parameter requires enabling multi-spindle control (setting bit 3 (MSP) of parameter No. 8133).

	#7	#6	#5	#4	#3	#2	#1	#0
3703					MPP			2P2
					MPP			

[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 2P2 When a 2-path system is used, inter-path spindle control allows:
0: Configuration where the spindle that belongs to one path only is shared between path 1 and path 2.
1: Configuration where the spindles that belong to path 1 and 2 are shared between the two paths.

#3 MPP In multi-spindle control, a spindle selection using a programmed command instead of using the signals (SWS1 to SWS2<G027.0, 1>) is:
0: Not made.
1: Made.

NOTE
When this parameter is set to 1, set parameter No. 3781 at the same time.

	#7	#6	#5	#4	#3	#2	#1	#0
3706					PCS			

[Input type] Parameter input
[Data type] Bit path

#3 PCS When multi-spindle control is enabled in each path of 2-path system, position coder selection signal (PC2SLC<Gn028.7>) is:

0: The signal on the side of the path selected by the inter-path spindle feedback signal is used.

1: The signal on the side of the local path is used.

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.

NOTE

1 This parameter is valid if bit 3 (MPP) of parameter No. 3703 is set to 1.
2 If this parameter is set to 0, the corresponding spindle cannot be selected by a P code.
3 Under 2-path control, the P code specified here is valid for each path.
For instance, if the P code to select the first spindle of path 2 is set to 21, specifying S1000 P21; in path 1 causes the first spindle of path 2 to be rotated at S1000.
4 Identical P code values cannot be used for different spindles. (Identical P code values cannot be used even if the paths are different.)
5 When this parameter is used (when bit 3 (MPP) of parameter No. 3703 is set to 1), the spindle command selection signal is invalid.
6 To use this parameter, enable multi-spindle control (bit 3 (MSP) of parameter No. 8133 is 1).

	#7	#6	#5	#4	#3	#2	#1	#0
8133					MSP			

NOTE

When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#3 MSP Multi-spindle is:

0: Not Used.
1: Used.

Alarm and message

Number	Message	Description
PS5305	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 parameter EMS (No. 3702#1) 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.

Note

NOTE

Signals used to operate the spindle control unit are not influenced by path spindle command selection signals; process them with a PMC Ladder program, if required.

(Example: SFRA<G070.5> is always a forward rotation command to the first serial spindle control amplifier of path 1.)

Reference item

Manual name	Item name
OPERATOR'S MANUAL (Lathe system) (B-64304EN-1)	Path spindle control

8.9 MEMORY COMMON TO PATHS

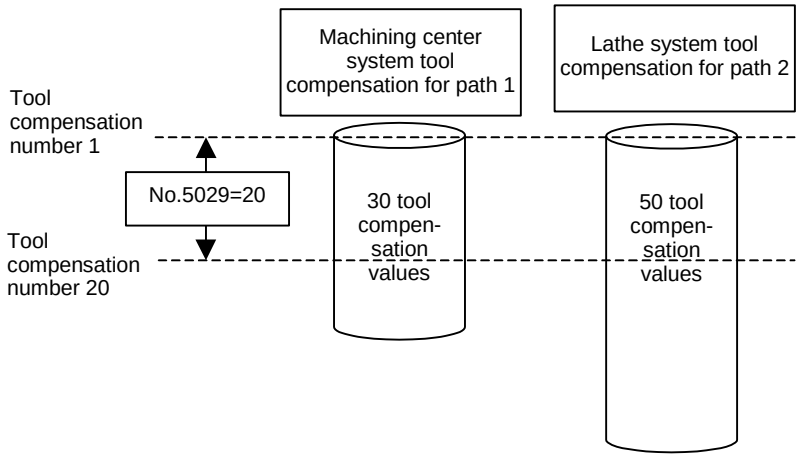
Overview

In a 2-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

- Tool compensation memory

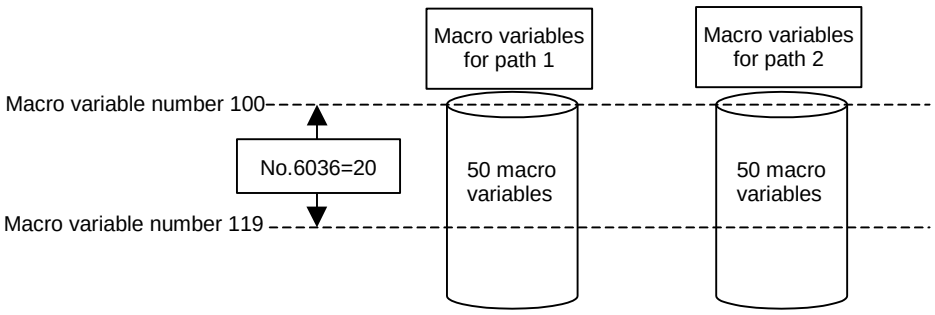
Part or all of tool compensation memory for individual paths can be used as common data by setting parameter No. 5029.



NOTE
1 Set a value less than the number of tool compensation values for each path for parameter No. 5029.
2 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.

- Custom macro common variables

All or part of custom macro common variables #100 to #149 (or #199) and #500 to #599 (or #999) can be used as common data by setting parameters Nos. 6036 (#100 to #199) and 6037 (#500 to #999).



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 If 0 or a negative value is set, the memory common to path is not used.

Parameter

5024	Number of tool compensation values

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 number of tool compensation values

Set the maximum allowable number of tool compensation values used for each path.
Ensure that the total number of values set in parameter No. 5024 for the individual paths is within the number of compensation values usable in the entire system.
If the total number of values set in parameter No. 5024 for the individual paths exceeds the number of compensation values usable in the entire system, or 0 is set in parameter No. 5024 for all paths, the number of compensation values usable for each path is a value obtained by dividing the number of compensation values usable in the entire system by the number of paths.
Tool compensation values as many as the number of compensation values used for each path are displayed on the screen. If tool compensation numbers more than the number of compensation values usable for each path are specified, an alarm is issued.
For example, 200 tool compensation sets are used, 120 sets may be allocated to path 1 and 80 sets to path 2. All of 200 sets need not be used.

5029	Number of tool compensation value memories common to paths

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 number of tool compensation values

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 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.

NOTE
2 When 0 or a negative value is set, memories common to paths are not used.

6036	Number of custom macro variables common to tool path (for #100 to #199)

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 100

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 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 #199: Used by each path independently

NOTE
When 0 or a negative value is set, the memory common to paths is not used.

6037	Number of custom macro variables common to tool path (for #500 to #999)

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 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
When 0 or a negative value is set, the memory common to paths is not used.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (Lathe system) (B-64304EN-1)	Memory common to paths

8.10 PATH SINGLE BLOCK CHECK FUNCTION

Overview

In 2-path control, each of the paths has single block command signals, so that their respective automatic operation programs can be brought to a single block stop. Even if, however, the single block command signals of the respective paths are set to "1", the times at which the programs will stop may not match, depending on the difference among command programs.

Example

Assuming that the following programs are started to perform SBK operation at the position of X0.0 in both paths

```
Path 1                  Path 2
O0001;                  O0002;
N1 G01 X10. F100 ;      N1 G01 X20. F100 ;
N2 X20.;                   :
   :                       :
```

In the above example, path 1 enters the single block stop state after the completion of the movement by "X10.", while in path 2, the movement by "X20." will be executed immediately.
This function prevents such an event from occurring; as soon as one path enters the single block stop state, this function places other paths in the feedhold stop states. This function enables single block operation while making sure that the machining programs of multiple paths have nearly been synchronized.
Because the single block command signals of the individual paths remain effective, so that uses such as the following are possible:

<1> Single block operation in which the single block command signal of path 1 only is set to "1" to follow the program of path 1
<2> Single block operation in which the single block command signals of multiple paths are set to "1" to follow the end of a specified block in one of the paths.

⚠ WARNING

If one of the paths is in the single block prohibited state or in the feedhold prohibited state due to threading, etc, the paths will not enter the stop state until that state is canceled. However, the paths enter the stop state if the single block prohibited state is set by the custom macro system variable #3003.

⚠ CAUTION

No operations such as synchronous processing on paths are performed. In an example such as that shown above, therefore, path 2 enters the feedhold stop state after the completion of the movement by "X10.", but the stop position will be near "X10.".

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8100		DSB						

[Input type] Parameter input

[Data type] Bit machine group

#6 DSB The inter-path single block check function is:
0: Disabled.
If one of the paths undergoes a single block stop, only the path undergoes a single block stop.
1: Enabled.
If one of the paths undergoes a single block stop, the other path undergoes a feed hold stop.

8.11 PATH SELECTION/DISPLAY OF OPTIONAL PATH NAMES

Overview

Path selection specifies whether operations performed using the MDI panel are for each path.
The operations, as used here, include displaying and setting data items (such as tool compensation values), entering command programs in MDI mode, and editing machining programs in program memory.
Also note that the name of each path can be changed by parameter.

Signal

Path selection signals (Tool post selection signal) HEAD<G063.0>

[Classification] Input signal
[Function] These signals select the path the MDI panel is to be for.
[Operation] The MDI panel operation is intended for the path specified by HEAD. The relationship between the path selection signal and the path to be selected is as follows.

Path selection signal HEAD<G063.0>	Path to be selected
0	Path 1
1	Path 2

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G063								HEAD

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8100							IAL	RST

[Input type] Parameter input
[Data type] Bit machine group

#0 RST The pressing of the reset key on the MDI panel is:
0: Valid for two paths.
1: Valid only for the path selected by the path selection signal.

#1 IAL Choice of an option concerning operation continuation when an alarm is issued, and choice of an option concerning the start of automatic operation in alarm state:
0:
- When an alarm is issued, the operation is stopped with the other path(s) in same group placed in hold state.
- When the other path or paths in same group are placed in alarm state, automatic operation cannot be started.

1: • Even when an alarm is issued, the operation is continued without stopping the other path(s).
• Even when the other path or paths in same group are placed in alarm state, automatic operation can be started.

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.

When 0 is set in parameter No. 3141, the path name is displayed according to the following table.

Language to display in CNC	Path name
English	HEAD1 (HEAD2)
Japanese	刃物台 1 (刃物台 2)
German	KANAL1 (KANAL2)
French	TETE1 (TETE2)
Traditional Chinese	HEAD1 (HEAD2)
Simplified Chinese	路径 1 (路径 2)
Italian	TEST1 (TEST2)
Korean	HEAD1 (HEAD2)
Spanish	CAB.1 (CAB.2)
Dutch	KAN.1 (KAN.2)
Danish	HOVED1 (HOVED2)

Language to display in CNC	Path name
Portuguese	CABEC.1 (CABEC.2))
Polish	GLOWIC1 (GLOWIC2)
Hungarian	FEJ1 (FEJ2)
Swedish	HUVUD1 (HUVUD2)
Czech	KANAL1 (KANAL2)
Russian	П У Т Ь 1 (П У Т Ь 2)
Turkish	HEAD1 (HEAD2)
Bulgarian	Г Л В А 1 (Г Л В А 2)

	#7	#6	#5	#4	#3	#2	#1	#0
3208			PSC					

[Input type] Setting input
[Data type] Bit

#5 PSC When the path is switched based on the path switch signal:
0: The screen display is switched to the last selected screen of the path.
1: The same screen as for the path before switching is displayed.

9 AUXILIARY FUNCTION

Chapter 9, "AUXILIARY FUNCTION", consists of the following sections:

9.1 AUXILIARY FUNCTION/2ND AUXILIARY FUNCTION 682
9.2 AUXILIARY FUNCTION LOCK.....694
9.3 MULTIPLE M COMMANDS IN A SINGLE BLOCK 695
9.4 HIGH-SPEED M/S/T/B INTERFACE 697

9.1 AUXILIARY FUNCTION/2ND AUXILIARY FUNCTION

Overview

- Auxiliary function (M code)

When an M code address is programmed, a 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. In some cases, however, up to three M codes can be specified for some types of machine tools (see "MULTIPLE M COMMANDS IN ONE BLOCK")
Also, parameter No. 3030 can specify the maximum digits and if a specified value exceeds the maximum digits, an alarm may be issued.

- 2nd auxiliary function (B code)

When up to eight digits are specified after address B, a code signal and strobe signal are sent. These signals are used to index the rotation axis of the machine. The code signal is retained until another B code is specified.
In each block, a single B code can be specified. The maximum number of digits that can be specified after address B is specified in parameter No. 3033. If a specified value exceeds the maximum digits, an alarm may be issued.
The address for specifying the 2nd auxiliary function can be changed from B to another address (A, C, U, V, or W) by parameter setting (parameter No. 3460).

Explanation

- Command format of 2nd auxiliary function

Command range

-99999999 to 99999999

Command method

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 parameter AUP (No.3450#0) 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 (parameter DPI (No.3401#0)).

Example:	Specified value	Output value
	B10	10

   When the second auxiliary function with a decimal point is specified, alarm PS0007 is issued.
   When the second auxiliary function is specified with a negative value, alarm PS0006 is issued.

2. When a command with a decimal point or a negative command is enabled
(When parameter AUP (No.3450#0) is set to 1)

When the desktop calculator decimal point setting is not specified (when parameter DPI (No.3401#0) 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 parameter DPI (No.3401#0) 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 9.1 (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 9.1 (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 parameter AUX (No.3405#0).

Table 9.1 (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		Parameter AUX = 0	Parameter AUX = 1
Metric input system	Reference axis: IS-A	100×	100×
	Reference axis: IS-B	1000×	1000×
	Reference axis: IS-C	10000×	10000×
Inch input system	Reference axis: IS-A	100×	1000×
	Reference axis: IS-B	1000×	10000×
	Reference axis: IS-C	10000×	100000×

⚠ CAUTION

If a decimal fraction remains after multiplying the specified value with a decimal point by a magnitude in Table 9.1 (a), the fraction is truncated.

Example:

Specified value	Output value	
B0.12345	1234	(Inch input system; the reference axis is IS-B; 1 is set in bit 0 (AUX) of parameter No. 3405. 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 is issued.
When the specified value is multiplied by a magnitude in Table 9.1 (a), the allowable number of digits must be set for the resultant value.

Basic procedure

The following signals are used with these functions. (For details of the spindle-speed function and tool function, see Chapters “SPINDLE SPEED FUNCTION” and “TOOL FUNCTION”.)

Function	Program address	Output signal			Input signal
		Code signal	Strobe signal	Distribution end signal	Completion signal
Auxiliary function	M	M00 to M31	MF	DEN	FIN
Spindle speed function	S	S00 to S31	SF		
Tool function	T	T00 to T31	TF		
2nd auxiliary function	B	B00 to B31	BF		

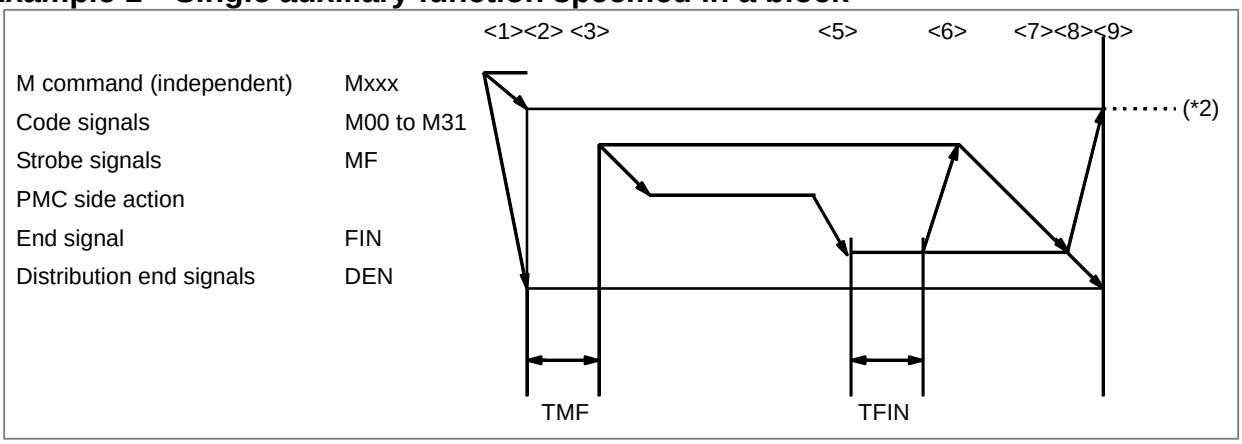
Each function uses different program addresses and different signals, but they all input and output signals in the same way, as described below. (A sample procedure for the auxiliary function is described below. The procedures for the spindle speed function, tool function, and 2nd auxiliary function, are obtained simply by substituting S, T, or B in place of M.)

<1> Suppose that Mxxx is specified in a program.
For xxx, the number of specifiable digits is specified in parameter Nos. 3030 to 3033 for each function. If a specified value exceeds the maximum digits, an alarm may be issued.
<2> Code signal M00 to M31 is sent to machine interface. After period TMF, specified in parameter No. 3010 (standard value: 16 msec), the strobe signal MF is set to 1. The code signal is the binary representation of the programmed value xxx.(*1) If a move, dwell, spindle, or other function is specified in the same block as the auxiliary function, the execution of the other function is started when the code signal of the auxiliary function is sent.
<3> When the strobe signal is set to 1, read the code signal and perform the corresponding operation on the PMC side.
<4> To execute an operation after the completion of the move, dwell or other function specified in the block, wait until distribution end signal DEN is set to 1.
<5> Upon completion of the operation, set completion signal FIN to 1 on the PMC side. The completion signal is used by the auxiliary function, spindle function, tool function, 2nd auxiliary function, external operation function described later, and other functions. If any of these functions are executed simultaneously, the completion signal must be set to 1 upon completion of all the functions.
<6> If the completion signal remains set to 1 for longer than period TFIN, specified in parameter No. 3011 (standard value: 16 msec), the CNC sets the strobe signal to 0 and reports that the completion signal has been received.
<7> When the strobe signal is set to 0, set the completion signal to 0 in the PMC.
<8> When the completion signal is set to 0, the CNC sets all code signals to 0 and completes all sequences of the auxiliary function.(*2)
<9> Once all other commands in the same block have been completed, the CNC executes the next block.

(*1) When the tool function is executed, the programmed tool number is sent as the code signal (T series).
(*2) When the spindle-speed function, tool function, or 2nd auxiliary function is executed, the code signal is maintained until a new code for the corresponding function is specified.

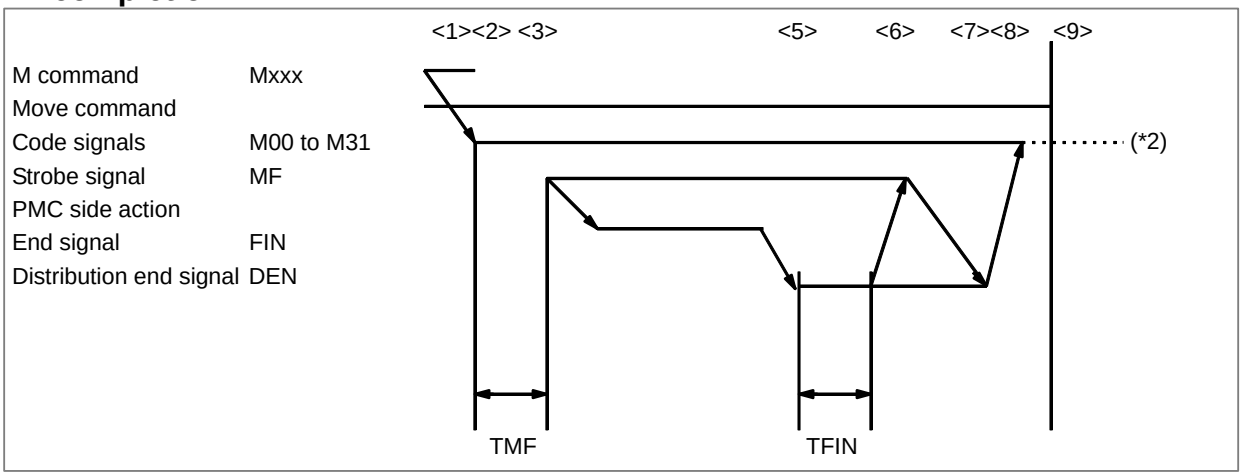
The timing diagram is shown below:

Example 1 Single auxiliary function specified in a block

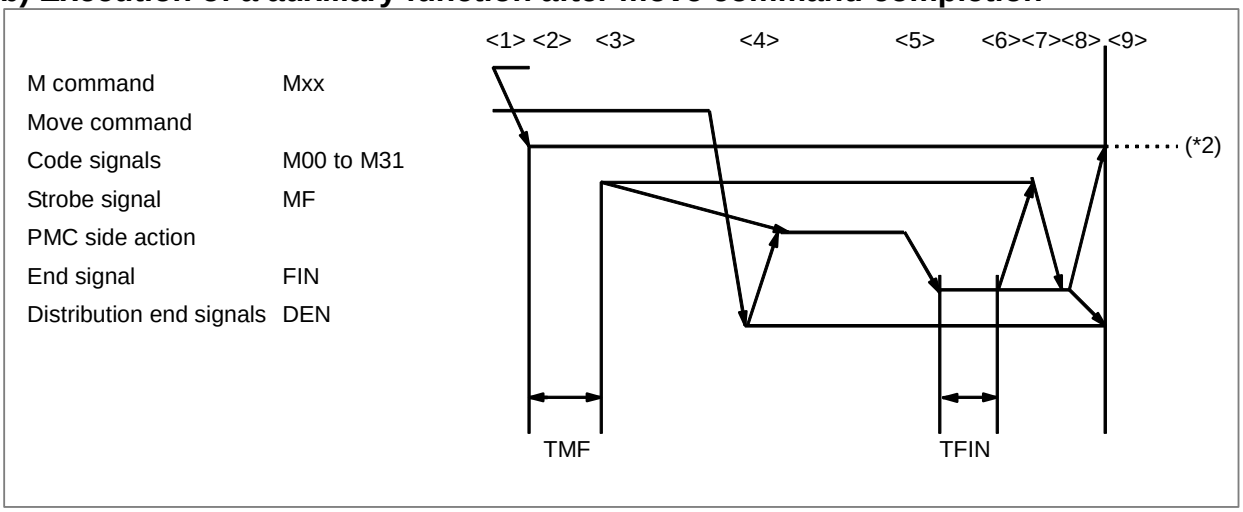


Example 2 Move command and auxiliary function in the same block

a) Execution of a auxiliary function without waiting for move command completion



b) Execution of a auxiliary function after move command completion



Signal

End signal FIN<Gn004.3>

[Classification] Input signal

[Function] This signal reports the completion of a auxiliary function, spindle function, tool function, 2nd auxiliary function, or external operation function.

[Operation] For the control unit operation and procedure when this signal goes "1", see the description of "Basic procedure" above.
The FIN signal must remain "1" for a certain time (TFIN, which is set by a parameter No. 3011) or longer. The FIN signal driven "1" is ignored if the FIN signal goes "0" before TFIN elapses.

> ⚠ **CAUTION**
> Only one end signal is used for all functions above. The end signal must go "1" after all functions are completed.

2nd auxiliary function end signal BFIN<Gn005.7>

[Classification] Input signal

[Function] This signal reports the completion of a 2nd auxiliary function using the high-speed M/S/T/B interface.

[Operation] For the control unit operation and procedure when this signal goes "1", see the description of "Basic procedure" above.

Distribution end signal DEN<Fn001.3>

[Classification] Output signal

[Function] These signals report that all commands (such as move commands and dwell) are completed except those auxiliary functions, spindle functions, 2nd auxiliary functions tool functions, and so forth that are contained in the same block and have been sent to the PMC. They also report that the end signal from the PMC is being awaited.

[Output cond.] The DEN signal turns to "1" when:

- Waiting for the completion of auxiliary functions, spindle functions, tool functions, and 2nd auxiliary functions, and all other commands in the same block are completed, and the current position is in the in-position.

The DEN signal turns to "0" when:

- The execution of one block is completed

> **NOTE**
> A parameter NCI (No.1601#5) can specify, whether to only check if an acceleration/deceleration delay is eliminated, or to also check if a servo delay (error) has been reduced to within a certain range.

Decode M signals DM00<Fn009.7>,DM01<Fn009.6>,DM02<Fn009.5>,DM30<Fn009.4>

[Classification] Output signal

[Function] These signals report particular auxiliary functions are specified. The auxiliary functions in a command program correspond to output signals as indicated below.

Command program	Output signal
M00	DM00
M01	DM01
M02	DM02
M30	DM30

[Output cond.] A decode M signal goes "1" when:
- The corresponding auxiliary function is specified, and any move commands and dwell commands specified in the same block are completed. These signals are not output when the end signal of the auxiliary function is returned before completion of such move commands and dwell commands.

A decode M signal goes "0" when:
- The FIN signal goes "1"
- Reset occurs

Auxiliary function code signals M00 to M31<Fn010 to Fn013>
Auxiliary function strobe signal MF<Fn007.0>

[Classification] Output signal
[Function] These signals report the specification of auxiliary functions.
[Output cond.] For the output conditions and procedure, see the description of "Basic procedure" above.

NOTE
1 The following auxiliary functions are only processed internally by the control unit; they are not output to the PMC when programmed:
- M98, M99, M198
- M code that calls a sub program (parameter No. 6071 to 6079)
- M code that calls a custom macro (parameter No. 6080 to 6089)

2 Decode signals as well as the code signals and strobe signal are output for the auxiliary functions listed below.
M00, M01, M02, M30

Spindle function code signals S00 to S31<Fn022 to Fn025>
Spindle function strobe signal SF<Fn007.2>

[Classification] Output signal
[Function] These signals report that spindle functions have been specified.
[Output cond.] For the output conditions and procedure, see the description of "Basic procedure" above.

Tool function code signals T00 to T31<Fn026 to Fn029>
Tool function strobe signal TF<Fn007.3>

[Classification] Output signal
[Function] These signals report that tool functions have been specified.
[Output cond.] For the output conditions and procedure, see the description of "Basic procedure" above.

2nd auxiliary function code signals B00 to B31<Fn030 to Fn033>
2nd auxiliary function strobe signal BF<Fn007.7>

[Classification] Output signal
[Function] These signals report that second auxiliary functions have been specified.
[Output cond.] For the output conditions and procedure, see the description of "Basic procedure" above.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn004					FIN			
Gn005	BFIN							

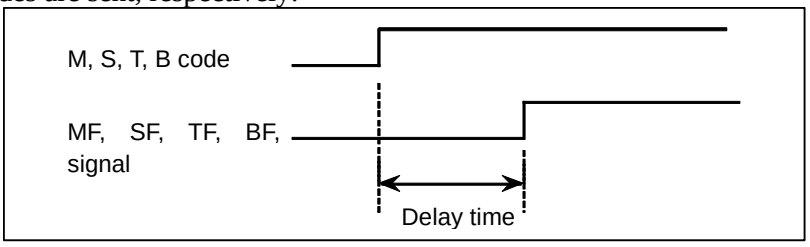
	#7	#6	#5	#4	#3	#2	#1	#0
Fn001					DEN			
Fn007	BF				TF	SF		MF
Fn009	DM00	DM01	DM02	DM30				
Fn010	M07	M06	M05	M04	M03	M02	M01	M00
Fn011	M15	M14	M13	M12	M11	M10	M09	M08
Fn012	M23	M22	M21	M20	M19	M18	M17	M16
Fn013	M31	M30	M29	M28	M27	M26	M25	M24
Fn022	S07	S06	S05	S04	S03	S02	S01	S00
Fn023	S15	S14	S13	S12	S11	S10	S09	S08
Fn024	S23	S22	S21	S20	S19	S18	S17	S16
Fn025	S31	S30	S29	S28	S27	S26	S25	S24
Fn026	T07	T06	T05	T04	T03	T02	T01	T00
Fn027	T15	T14	T13	T12	T11	T10	T09	T08
Fn028	T23	T22	T21	T20	T19	T18	T17	T16
Fn029	T31	T30	T29	T28	T27	T26	T25	T24
Fn030	B07	B06	B05	B04	B03	B02	B01	B00
Fn031	B15	B14	B13	B12	B11	B10	B09	B08
Fn032	B23	B22	B21	B20	B19	B18	B17	B16
Fn033	B31	B30	B29	B28	B27	B26	B25	B24

Parameter

3010	Time lag in strobe signals MF, SF, TF, and BF

[Input type] Parameter input
[Data type] Word path
[Unit of data] msec
[Valid data range] 0 to 32767

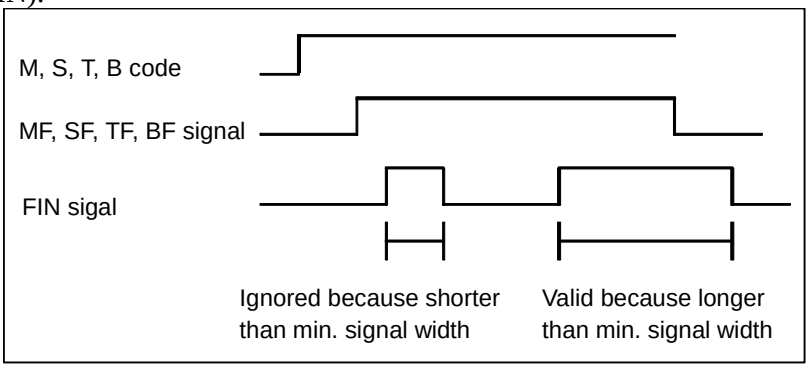
The time required to send strobe signals MF, SF, TF, and BF after the M, S, T, and B codes are sent, respectively.



NOTE
The time is counted in units of 8 ms. If the set value is not a multiple of four, it is raised to the next multiple of eight
Example
When 30 is set, 32 ms is assumed.
When 0 is set, 8 ms is assumed.
The time count period may change, depending on the system.

3011	Acceptable width of M, S, T, and B function completion signal (FIN)

[Input type] Parameter input
[Data type] Word path
[Unit of data] msec
[Valid data range] 0 to 32767
Set the minimum signal width of the valid M, S, T, and B function completion signal (FIN).



NOTE
The time is counted in units of 8 ms. If the set value is not a multiple of four, it is raised to the next multiple of eight
Example
When 30 is set, 32 ms is assumed.
When 0 is set, 8 ms is assumed.
The time count period may change, depending on the system.

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.

NOTE
The allowable number of digits for the S code is 1 to 5 digits. If 0 is set in parameter No. 3031, the allowable number of digits is assumed to be 5.

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) is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
3401								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)

	#7	#6	#5	#4	#3	#2	#1	#0
3404			M02	M30				

[Input type] Parameter input
[Data type] Bit path

#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 end signal FIN is returned according to M30 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 according to M02 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	#6	#5	#4	#3	#2	#1	#0
3405								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
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

3411	**M code preventing buffering (1)**
to	**to**
3420	**M code preventing buffering (10)**

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 999999999

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 parameter 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.
performed until the execution of the block is completed.

NOTE
1 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.
2 If the minimum value is greater than the maximum value, the setting is invalid.
3 If there is only one data item, the minimum value must be equal to the maximum value.

	#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) is issued.
3. When a negative value is specified
The alarm (PS0006) 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.)

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.

Note

NOTE

1 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 sequence of PMC.
2 When the 2nd auxiliary function is provided, the address used for specifying the 2nd auxiliary function (B or the address specified with parameter No. 3460) cannot be used as an axis address.
3 The block following M00, M01, M02 and M30, is not read into the input buffer register, if present. Similarly, ten M codes which do not buffer can be set by parameters (No. 3411 to 3420).
4 For M00 and M01, auxiliary function code signal, auxiliary function strobe signal, and M decode signals are only sent; the control of program stop and optional stop shall be designed on the PMC side.
5 When the automatic operation is stopped by M02 or M30, it is necessary to send the external reset signal from the machine side to the CNC, instead of the FIN signal. When the external reset signal is returned against the M02 or M30, the control returns to the top of the program recently executed, and enters the reset state. When the FIN signal is returned, the control returns to the beginning of the program recently executed, and executes it from the top.

Alarm and message

Number	Message	Description
PS0003	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.
PS0006	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.
PS0007	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.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Auxiliary function (M code)
	2nd auxiliary function (B code)

9.2 AUXILIARY FUNCTION LOCK

Overview

Inhibits execution of a specified M, S, T and B function. That is, code signals and strobe signals are not issued. This function is used to check a program.

Signal

Auxiliary function lock signal AFL<Gn005.6>

[Classification] Input signal

[Function] This signal selects auxiliary function lock. That is, this signal disables the execution of specified M, S, T, and B functions.

[Operation] When this signal turns to "1", the control unit functions as described below.

(1) The control unit does not execute M, S, T, and B functions specified for memory operation, DNC operation, or MDI operation. That is, the control unit stops the output of code signals and strobe signals (MF, SF, TF, BF).

(2) If this signal turns to "1" after code signal output, the output operation is executed in the ordinary manner until its completion (that is, until the FIN signal is received, and the strobe signal turns to "0").

(3) Among the auxiliary functions, M00, M01, M02, and M30 are executed even when this signal is "1". All code signals, strobe signals, decode signals are output in the ordinary manner.

(4) Among the auxiliary functions, even when this signal is "1", those functions (M98 and M99) that are executed in the control unit without outputting their execution results are executed in the ordinary manner.

> ⚠ **CAUTION**
> Even when this signal is "1", spindle analog output or spindle serial output is executed.

Auxiliary function lock check signal MAFL<Fn004.4>

[Classification] Output signal

[Function] This signal reports the state of the auxiliary function lock signal AFL.

[Output cond.] This signal turns to "1" when:

- The auxiliary function lock signal AFL is "1"

This signal turns to "0" when:

- The auxiliary function lock signal AFL is "0"

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn005		AFL						

	#7	#6	#5	#4	#3	#2	#1	#0
Fn004				MAFL				

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Machine lock and auxiliary function lock

9.3 MULTIPLE M COMMANDS IN A SINGLE BLOCK

Overview

So far, one block has been able to contain only one M code. However, this function allows up to three M codes to be contained in one block.
Up to three M codes specified in a block are simultaneously output to the machine (PMC). This means that compared with the conventional method of a single M command in a single block, a shorter cycle time can be realized in machining.

(Example)

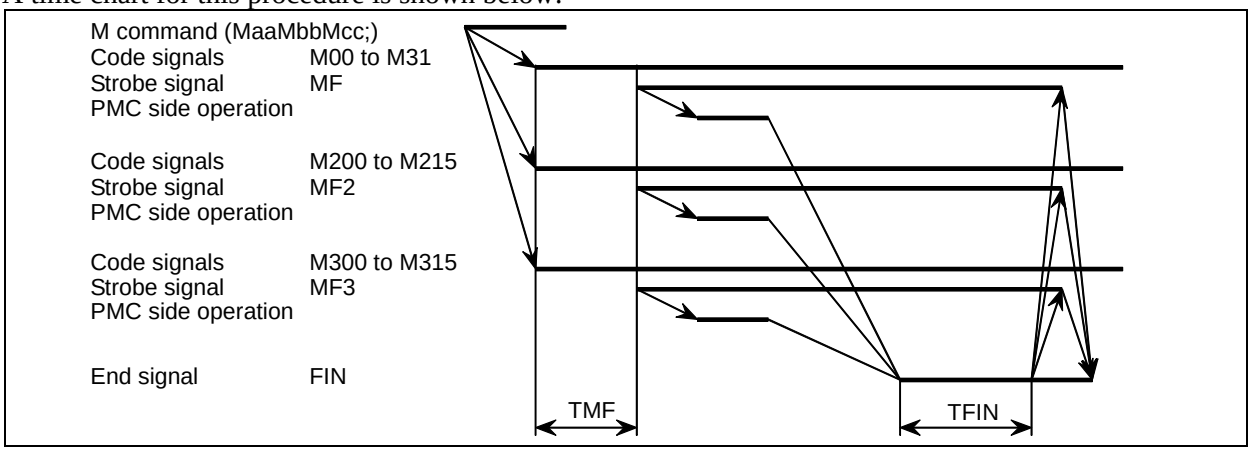
One M command in a single block	Multiple M commands in a single block
M40; M50; M60; G28G91X0Y0Z0; : : :	M40M50M60; G28G91X0Y0Z0; : : : : :

Explanation

Basic procedure

(1) Assume that "MaaMbbMcc;" was commanded by the program.
(2) The 1st M command (Maa) sends the code signals M00 to M31 in a manner similar to the conventional one-block single command. The strobe signal MF is set to "1" after a time TMF set by parameter No. 3010 (Standard setting: 16 msec).
The second M command (Mbb) sends the code signal M200-M215, the third M command (Mcc) sends the code signal M300-M315, and their respective strobe signals MF2 and MF3 are set to "1".
Furthermore, the three code signals are sent simultaneously.
The strobe signals MF, MF2, and MF3 become "1" at the same time.
The code signal is a binary notation of the program command aa, bb and cc.
(3) On the PMC side, read the code signals corresponding to the respective strobe signals when the strobe signals become "1", and perform the appropriate operations.
(4) When the operation of all M commands ends on the PMC side, the end signal (FIN) is set to "1".
(5) When the completion signal stays "1" for a time (TFIN) set by parameter No. 3011 (Standard: 16 msec), all strobe signals (MF, MF2 and MF3) are set to "0" at the same time and the reception of completion signal is reported.
(6) On the PMC side, when MF, MF2 and MF3 are set to "0", the completion signal is set to "0".

A time chart for this procedure is shown below:



Signal

2nd, 3rd M function code signals
M200 to M215<Fn014 to Fn015>,M300 to M315<Fn016 to Fn017>
2nd, 3rd M Function strobe signals MF2 <Fn008.4>,MF3<Fn008.5>

[Classification] Output signal

[Function] Indicates that second and third auxiliary functions have been issued.

[Output cond.] For the output conditions and procedure, see the description of “Basic Procedure” above.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn008			MF3	MF2				
Fn014	M207	M206	M205	M204	M203	M202	M201	M200
Fn015	M215	M214	M213	M212	M211	M210	M209	M208
Fn016	M307	M306	M305	M304	M303	M302	M301	M300
Fn017	M315	M314	M313	M312	M311	M310	M309	M308

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3404	M3B							

[Input type] Parameter input

[Data type] Bit path

#7 M3B The number of M codes that can be specified in one block

0: One

1: Up to three

Caution

⚠ CAUTION

1 M00, M01, M02, M30, M98, M99, or M198 must not be specified together with another M code.

2 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 alone in a single block. Such M codes include those that make the CNC perform internal operations in addition to sending the M code signals to the PMC. For example, such M codes are M codes for calling program numbers 9001 to 9009 and M codes for disabling advance reading (buffering) of subsequent blocks.
The M codes that can be specified in a single block must be those that the CNC only sends the M code signals to the PMC side.

Note

NOTE

1 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 example, M42 can be specified only after the mechanical operation of M41 is completed.

NOTE
2 The 1st M code can be up to 8 digits and 2nd, 3rd M codes can be the values up to 65535.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Multiple M commands in a single block

9.4 HIGH-SPEED M/S/T/B INTERFACE

Overview

To accelerate M/S/T/B function execution, the high-speed M/S/T/B interface has simplified the transfer of the strobe and completion signals of the M/S/T/B functions.
Whether to use the usual system or high-speed system for strobe signal and completion signal handling can be specified by parameter MHI (No. 3001#7).
The description below uses the auxiliary functions (M code commands) as an example. The same description applies to the spindle function (S code), tool function (T code) and 2nd auxiliary function (B code).

Explanation

- Basic procedure

(1) Assume that the following program is given:
Mxx;
Myy;

(2) In response to an M command, the CNC system sends out the code signals M00 to M31.
The CNC system inverts the logical level of the strobe signal MF, that is, from "0"to "1", or from "1" to "0".

(3) The CNC system inverts the strobe signal, then when the logical level of the auxiliary function completion signal MFIN becomes the same as the strobe signal, the CNC assumes the completion of PMC sequence.
With the usual method, the operation is assumed to be completed when a falling edge ("1" to "0") of the M/S/T/B completion signal FIN is received after a rising edge ("0" to "1") of the FIN signal is detected. This new system, on the other hand, assumes the operation has been completed upon detection of only one transition of the completion signal.
In addition , the usual system uses only one completion signal (FIN) common to the M/S/T/B functions. This new system uses a different completion signal for each of the M, S, T, and B functions; the completion signals for the M, S, T, and B functions are MFIN, SFIN, TFIN, and BFIN, respectively.
The Fig. 9.4 (a) below shows the timing chart of these signals with the new system. For comparison, Fig. 9.4 (b) shows the timing chart of the conventional system.

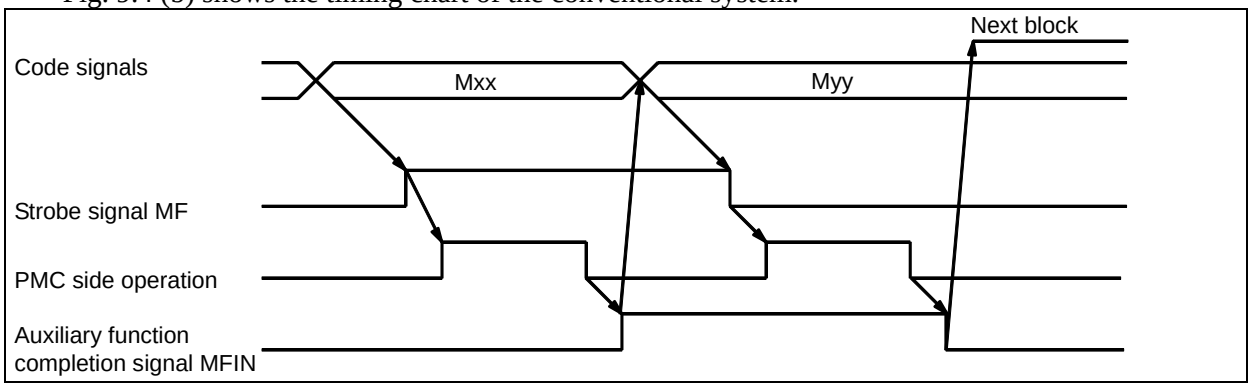


Fig. 9.4 (a) Timing chart of the high-speed system

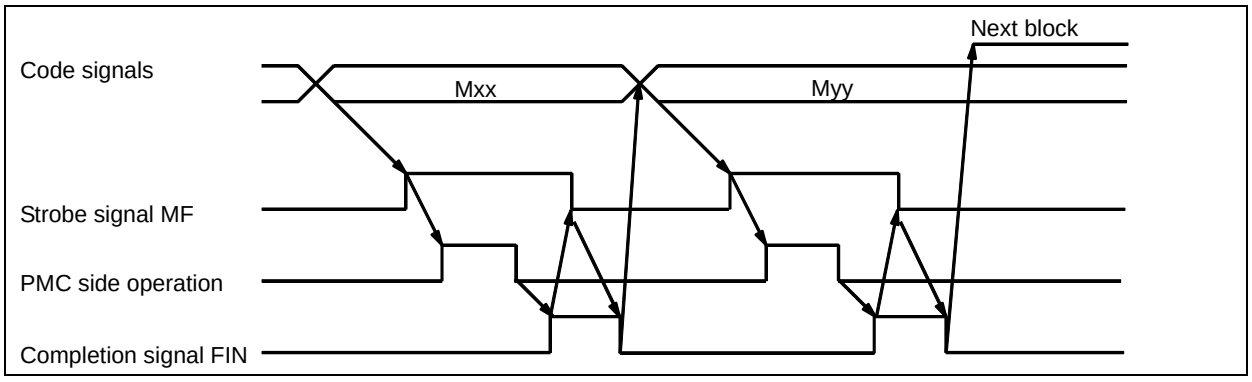


Fig. 9.4 (b) Timing chart of the usual system

A high-speed interface can also be used for multiple M commands issued for one block. This interface provides separate completion signals for each of the 3 M codes. They are called MFIN (the same name as for the single M command per block function), MFIN2, and MFIN3, respectively. The signal transfer sequence for multiple M codes per block is the same as that for a single M code per block.
The high-speed interface can also be used for the external operation function. In this case, special external operation signal EFD and completion signal EFIN are used. The procedure for sending and receiving these signals is identical to that for sending and receiving the strobe and completion signals of the auxiliary function (M series).

Signal

Auxiliary function completion signal MFIN<Gn005.0>

[Classification] Input signal
[Function] Reports that the execution of a auxiliary function using the high-speed M/S/T/B interface is completed.
[Operation] For the operation and procedure of the control unit when this signal turns to "1" or "0", see the description of "Basic procedure" above.

Spindle function completion signal SFIN<Gn005.2>

[Classification] Input signal
[Function] Reports that the execution of a spindle speed function using the high-speed M/S/T/B interface is completed.
[Operation] For the operation and procedure of the control unit when this signal turns to "1" or "0", see the description of "Basic procedure" above.

Tool function completion signal TFIN<Gn005.3>

[Classification] Input signal
[Function] Reports that the execution of a tool function using the high-speed M/S/T/B interface is completed.
[Operation] For the operation and procedure of the control unit when this signal turns to "1" or "0", see the description of "Basic procedure" above.

2nd auxiliary function completion signal BFIN<Gn005.7>

[Classification] Input signal
[Function] Reports that the execution of a second auxiliary function using the high-speed M/S/T/B interface is completed.
[Operation] For the operation and procedure of the control unit when this signal turns to "1" or "0", see the description of "Basic procedure" above.

2nd, 3rd M function completion signals MFIN2<Gn004.4>,MFIN3<Gn004.5>

[Classification] Input signal

[Function] Indicate that when the high-speed interface is used for multiple M commands per block, the 2nd to 3rd M functions have been completed.

[Operation] For the operation and procedure of the control unit when this signal turns to "1" or "0", see the description of "Basic procedure" above.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn004			MFIN3	MFIN2				
Gn005	BFIN				TFIN	SFIN		MFIN

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3001	MHI							

[Input type] Parameter input

[Data type] Bit path

#7 MHI Transfer of strobe and completion signals for the M, S, T, and B

0: Normal

1: High-speed

	#7	#6	#5	#4	#3	#2	#1	#0
3002								CHM

[Input type] Parameter input

[Data type] Bit path

#0 CHM For high-speed M/S/T/B, the distribution end signal DEN <Fn001.3> and an auxiliary function code signal M00 to M31 <Fn010 to Fn013> are:

0: Not turned off even upon completion of the execution of the auxiliary function.

1: Turned off upon completion of the execution of the auxiliary function.

Note

NOTE

1 The strobe signals MF, SF, TF, and BF are "0" when the power is turned on.

2 When the control unit is reset, MF, SF, TF, and BF are set to "0".

3 When the execution of one block is completed, it is selected whether the distribution end signal DEN and an auxiliary function code signal M00 to M31 are set to 0 by bit 0 (CHM) of parameter No.3002.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION)	Auxiliary function, 2nd auxiliary function
(This manual)	Multiple M commands in a single block

10 SPINDLE SPEED FUNCTION

Chapter 10, "SPINDLE SPEED FUNCTION", consists of the following sections:

10.1 SPINDLE SPEED FUNCTION (S CODE OUTPUT)....700
10.2 DESIGNATION OF SPINDLE AXES....701
10.3 SPINDLE SERIAL OUTPUT....705
10.4 SPINDLE ANALOG OUTPUT....715
10.5 SERIAL/ANALOG SPINDLE CONTROL....719
10.6 SPINDLE SPEED CONTROL....722
10.7 SPINDLE OUTPUT CONTROL BY THE PMC....749
10.8 CONSTANT SURFACE SPEED CONTROL....755
10.9 ACTUAL SPINDLE SPEED OUTPUT (T SERIES)....763
10.10 SPINDLE POSITIONING (T SERIES)....764
10.11 Cs CONTOUR CONTROL....797
10.12 MULTI-SPINDLE CONTROL....826
10.13 RIGID TAPPING....843
10.14 SPINDLE SYNCHRONOUS CONTROL....927
10.15 SPINDLE ORIENTATION....945
10.16 SPINDLE OUTPUT SWITCHING....948
10.17 SIMPLE SPINDLE SYNCHRONOUS CONTROL (M SERIES)....949
10.18 SPINDLE SPEED FLUCTUATION DETECTION (T SERIES)....960
10.19 SPINDLE CONTROL WITH SERVO MOTOR....970

10.1 SPINDLE SPEED FUNCTION (S CODE OUTPUT)

Overview

When up to five digits are specified after address S, code and strobe signals are sent out and used to control the spindle speed. The code signals are retained until another S code is issued.
One S code is used for each block. Parameter No. 3031 can be used to specify the maximum number of digits. If a number greater than the maximum number is specified, an alarm can be raised.

Signal

Refer to section "Auxiliary Function/Second Auxiliary Function."

Parameter

3031	Allowable number of digits for the S code

[Data type] Byte
[Valid data range] 1 to 5
Set the allowable numbers of digits for the S code.

Note

NOTE
1 When a move command and a spindle-speed function command are specified within the same block, the commands are executed in either of the following two ways:
   (1) The move command and spindle-speed function are started at the same time.
   (2) After the move command is completed, the spindle-speed function command is started.
   Whether (1) or (2) takes place depends on the processing on the PMC side.
2 For S code output when spindle serial output or spindle analog output is used, see Section 10.6, "SPINDLE SPEED CONTROL".

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (B-64303EN-1) (this manual)	Auxiliary function/2nd auxiliary function
	Spindle speed control

10.2 DESIGNATION OF SPINDLE AXES

Overview

This section explains the designation of spindle axes and required settings.

Explanation

The following lists the number of controlled spindles in each model.

	Initial value	Max. Number of spindle in the path	Max. Number of spindle Total
Series 0*i*-MD	1	2	2
Series 0*i*-TD	1	3	3
Series 0*i*-TD (2-path control)	1	3	4(Total of two-path)
Series 0*i* Mate-MD	1	1	1
Series 0*i* Mate-TD	1	2	2

NOTE
1 The maximum numbers above vary with the option configuration. For details, refer to the manual provided by the machine tool builder.
2 The number of feed axes may limit the maximum number of spindle axes. (The limitation by the Total number of control axes)
See 1.1 "Controlled axis" for details.

- Setting of the number of spindles

- When serial spindle output is enabled (0 is set in bit 5 (SSN) of parameter No. 8133)
  Based on bit 1 (ISI) and bit 4 (SS2) and bit 5 (SS3) of path-related parameter No. 3701, the number of spindles in each path is set as follows.

SS3	SS2	ISI	Number of spindles in the path
"0"	"0"	"1"	0
"0"	"0"	"0"	1
"0"	"1"	"0"	2
"1"	"0"	"0"	3

- When serial spindle output is disabled (1 is set in bit 5 (SSN) of parameter No. 8133)
  One spindle is assigned to each path, and one analog spindle can be controlled. In this case, the spindle of the second path is not used; therefore, set 0 in parameter No. 3717 when setting the amplifier number of the spindle.
  Note that bit 1 (ISI) and bit 4 (SS2) and bit 5 (SS3) of parameter No. 3701 become invalid.

When spindle control with servo motor is enabled, the servo motor used as an axis subject to spindle control with servo motor is treated as a spindle, so set the number of spindles including this axis. The number of connectable spindles is reduced by the number of axes subject to spindle control with servo motor.
If the total number of spindles exceeds the maximum value, an alarm PS1996 occurs.

- Assignment of spindles to paths

You can specify which path each spindle is to be assigned, using parameter No. 982.
You can assign each path to any controlled axis, provided that you follow the maximum allowable number of spindles.
Example 1)
In the case of a 2-path control system with one spindle assigned to both path 1 and path 2

Parameter No. 0982 setting	Application
1	Path 1, spindle 1
2	Path 2, spindle 1

Example 2)
In the case of a 2-path control system with two spindles assigned to path 1 and no spindles assigned to path 2

Parameter No. 0982 setting	Application
1	Path 1, spindle 1
1	Path 1, spindle 2

NOTE
1 If the setting of parameter No. 982 is 0, that spindle will belong to the first path.
2 If the assignment of spindles to paths specified by parameter No. 0982 does not match the number of spindles set for each path by bit 1 (ISI) and bit 4 (SS2) of parameter No. 3701, an alarm PS0365 occurs.
3 If this alarm PS0365 occurs, the system comes to an emergency stop. In this case, set an appropriate value in parameter No. 0982 and then turn the power back on.

- Automatic setting

If all the values set in parameter No. 0982 are 0, the parameter is automatically set based on the number of spindles set for each path.

Parameter

0982	Absolute path number to which each spindle belongs

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] 1, 2
Set the path to which each spindle belongs.

NOTE
1 When 0 is set for all axes, the parameter is automatically set according to the number of controlled axes of each path.
2 When the setting falls outside the range, the axis is assumed to belong to the first path.
3 When spindle control with servo motor is enabled, the servo motor used as the spindle controlled axis is treated as a spindle. Therefore, it is necessary to set the path to which the axis subject to spindle control with servo motor.

	#7	#6	#5	#4	#3	#2	#1	#0
3701			SS3	SS2			ISI	

[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.

#1 ISI
#4 SS2
#5 SS3 Set the number of spindles in a path.

SS3	SS2	ISI	Number of spindles in a path
"0"	"0"	"1"	0
"0"	"0"	"0"	1
"0"	"1"	"0"	2
"1"	"0"	"0"	3

NOTE
This parameter is valid when spindle serial output is enabled (bit 5 (SSN) of parameter No. 8133 is 0).

	#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 To use a serial spindle, set bit 5 (SSN) of parameter No. 8133 to 0.
2 A maximum of one analog spindle can be controlled.
3 When using an analog spindle, set it at the end of the spindle configuration.

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.
3: Spindle motor connected to amplifier number 3 is used.

NOTE
When using an analog spindle, set it at the end of the spindle configuration.
(Example)
When there are three spindles in an entire system (two serial spindles and one analog spindle), set the spindle amplifier number (this parameter) of the analog spindle to 3.

	#7	#6	#5	#4	#3	#2	#1	#0
8133			SSN					

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#5 SSN Spindle serial output is:
0: Used.
1: Not Used.

Alarm and message

Number	Message	Description
PS0365	TOO MANY MAXIMUM SV/SP AXIS NUMBER PER PATH	The Max. total number of control axes, the Max. number of feed axes or the Max. number of control axes is exceeded. Check parameter No. 0981 and No. 0982. If this alarm is issued, emergency stop cannot be released.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (This manual)	Controlled axis
CONNECTION MANUAL (FUNCTION) (This manual)	2-path control

10.3 SPINDLE SERIAL OUTPUT

Overview

Spindle serial output is a spindle motor control interface.
The spindle serial output can control up to three serial spindles (up to two serial spindles per path).

NOTE
When a serial spindle is to be used, set bit 5 (SSN) of parameter No. 8133 to 0, and set bit 0 (A/S) of parameter No. 3716.

Explanation

The terms "logical n-th spindle", "n-th spindle in a path", and "n-th spindle amplifier" in the subsequent descriptions refer to the following:

- Logical n-th spindle:
  The n-th spindle to be controlled in the overall system
- N-th spindle (in a path):
  The n-th spindle to be controlled in a path.
  The first and second spindles in the descriptions refer to relative spindle numbers within a path. (For a single-path system, the logical n-th spindle equals the n-th spindle.)
- N-th spindle amplifier:
  The n-th spindle amplifier (amplifier No. n) actually connected to the CNC.

For the relationships between them, see "Spindle numbers vs. spindle motors" described later.

For how to set a spindle configuration for each path in multipath control, see the description of spindles in "2-path Control".
The table below lists the relationship between the spindles and functions.

Function \ Spindle	Serial spindle (*1)	
	First spindle	Second spindle
Threading/feed per revolution (synchronous feed)	Available	Available (*2)
Constant surface spindle control	Available	Available (*2)
Spindle speed fluctuation detection (T series)	Available	Available (*2)
Actual spindle speed output (T series)	Available	Available (*2)
Spindle positioning (T series)	Available	Available (*2)
Cs contour control	Available	Available
Multi-spindle	Available	Available
Rigid tapping	Available	Available (*2)
Spindle synchronous control	Available	Available

Function \ Spindle	Serial spindle (*1)	
	First spindle	Second spindle
Simple spindle synchronous control (M series)	Available	Available
Spindle control unit functions, such as spindle orientation, spindle output switching, spindle switching, and etc. (*3)	Available	Available
Polygon turning (using the servo motor axis and spindle)	Available	Available
Polygon machining with two spindles (T series) (Spindle and spindle polygon)	Available	Available
Spindle output control by the PMC	Available	Available

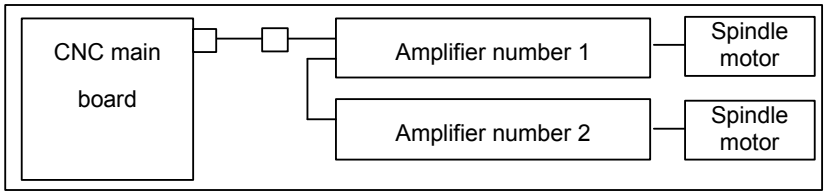
NOTE
1 The spindle serial output is required.
2 Multi-spindle control is required. (To use multi-spindle control, set bit 3 (MSP) of parameter No. 8133 to 1, and bit 1 (EMS) of parameter No. 3702 to 0.)
3 These functions belong to the spindle control unit. They cannot be used unless the spindle control unit supports those functions.

- Spindle numbers vs. spindle motors

Logical spindle numbers are associated with spindle motors as follows:

1. Logical spindle numbers and amplifier numbers
   Logical spindle numbers indicate the order of spindles to be controlled (the logical arrangement order). Parameters for the individual spindles are arranged in this order. Logical spindle numbers are assigned sequentially from 1 to the maximum number of spindles. DI/DO signals are allocated for each spindle number within a path, and their signal locations are incremented by 1000 for each path.
   Amplifier numbers indicate the order of spindle amplifiers actually connected to the CNC (the physical arrangement order). They are assigned as shown below. When a spindle motor is connected to each spindle amplifier, the spindle amplifier numbers and spindle motor numbers match.

2. Setting method



Connection of serial spindles

NOTE
In two-path control in the T series, three serial spindles can be used additionally by using the spindle serial output option (three spindles).

For each spindle, set the type of the motor in bit 0 (A/S) of parameter No. 3716, and set the amplifier number in parameter No. 3717. For spindles not to be used, set 0 as the amplifier numbers.

Example 1)
When using one serial spindle in a single-path system

Parameter	First spindle	Second spindle
A/S(No.3716#0)	1	0
No.3717	1	0

Example 2)
When using two serial spindles in a single-path system

Parameter	First spindle	Second spindle
A/S(No.3716#0)	1	1
No.3717	1	2

Signal

Spindle control unit signals for the serial spindle
<Gn070-Gn073><Gn304-Gn307><Fn045-Fn048><Fn306-Fn307> : First spindle
<Gn074-Gn077><Gn308-Gn311><Fn049-Fn052><Fn308-Fn309> : Second spindle

These addresses are assigned to locations on the CNC, but the signals at these addresses are input/output signals of the serial spindle control unit. For details of the signals at these addresses, refer to the following manuals of serial spindles:
FANUC SERVO AMPLIFIER α*i* series DESCRIPTIONS (B-65282EN)
FANUC AC SPINDLE MOTOR α*i* series DESCRIPTIONS (B-65272EN)
FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)

Spindle ready signals SRSP1R<Fn034.6> : First spindle
SRSP2R<Fn034.5> : Second spindle

[Classification] Output signal
[Function] These signals post that a corresponding spindle is ready for operation.
[Output cond.] These signals are set to 1 when:
The corresponding spindle becomes ready for operation (when the following conditions are met).
- Use of the corresponding spindle is specified.
- There is no serial communication alarm. (Serial spindle)
- Serial spindle parameters have been transferred. (Serial spindle)
- The spindle control software is ready for operation. (Serial spindle)

These signals are set to 0 when:
The corresponding spindle is not ready for operation.

All-spindle operation ready signal SRSRDY <F0034.7>

[Classification] Output signal
[Function] This signal posts that all spindles used are ready for operation.
[Output cond.] The signal is set to 1 when:
All spindles used become ready for operation.
The signal is set to 0 when:
Some of the spindles used are not ready for operation.

Spindle warning detail signals
SPWRN1 to SPWRN9 <Fn264.0 to Fn265.0>

[Classification] Output signal
[Function] Before an alarm is issued, the occurrence of a warning condition is posted via the warning interface, which is independent of the conventional alarm interface.
Nine signals are output to indicate a decimal code. The relationship between the spindle warning number and signals is expressed as follows:
[Output cond.] A warning number is output according to the spindle amplifier status as follows:

$$\text{Spindle warning No.} = \sum_{i=1}^{8} \left\{ 2^{i-1} \times \text{SPWRN}_i \right\} + 2^8 \times \text{SPWRN}_9$$

Warning No.	Description of warning	Spindle operation
56	SPM internal cooling fan stopped	When the stopped state of the internal cooling fan is detected, the warning number is posted from the spindle amplifier to the CNC. The spindle continues operating. About one minute later, an alarm number is posted, and the spindle is placed in the free running state.
88	SPM radiator cooling fan stopped	When the stopped state of the external cooling fan is detected, the warning number is posted from the spindle amplifier to the CNC. The spindle continues operating. If the main circuit overheats because the external cooling fan is stopped, an alarm is posted, and the spindle is placed in the free running state.
58	Overload on converter main circuit	A warning number is output according to the signal received from the PSM In the warning state, the spindle continues operating.
59	Converter internal cooling fan stopped	
113	Converter radiator cooling fan stopped	

The spindle warning detail signals are set to 0 when all alarm factors are eliminated.
When a warning is issued in more than one spindle within the same path, the spindle warning detail signals indicate the warning No. of the spindle having the smallest spindle number.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F0034	SRSRDY							
	#7	#6	#5	#4	#3	#2	#1	#0
Fn034		SRSP1R	SRSP2R					
	#7	#6	#5	#4	#3	#2	#1	#0
Fn264	SPWRN8	SPWRN7	SPWRN6	SPWRN5	SPWRN4	SPWRN3	SPWRN2	SPWRN1
Fn265								SPWRN9

- **Serial spindle control unit signals**
  For first spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn070	MRDYA	ORCMA	SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
Gn071	RCHA	RSLA	INTGA	SOCNA	MCFNA	SPSLA	*ESPA	ARSTA
Gn072	RCHHGA	MFNHGA	INCMDA	OVRA	DEFMDA	NRROA	ROTAA	INDXA
Gn073						MPOFA	SLVA	MORCMA
Gn304								
Gn305								
Gn306								
Gn307								
Fn045	ORARA	TLMA	LDT2A	LDT1A	SARA	SDTA	SSTA	ALMA
Fn046	MORA2A	MORA1A	PORA2A	SLVSA	RCFNA	RCHPA	CFINA	CHPA
Fn047							INCSTA	PC1DEA
Fn048								
Fn306								
Fn307								

For second spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn074	MRDYB	ORCMB	SFRB	SRVB	CTH1B	CTH2B	TLMHB	TLMLB
Gn075	RCHB	RSLB	INTGB	SOCNB	MCFNB	SPSLB	*ESPB	ARSTB
Gn076	RCHHGB	MFNHGB	INCMDB	OVRB	DEFMDB	NRROB	ROTAB	INDXB
Gn077						MPOFB	SLVB	MORCMB
Gn308								
Gn309								
Gn310								
Gn311								
Fn049	ORARB	TLMB	LDT2B	LDT1B	SARB	SDTB	SSTB	ALMB
Fn050	MORA2B	MORA1B	PORA2B	SLVSB	RCFNB	RCHPB	CFINB	CHPB
Fn051							INCSTB	PC1DEB
Fn052								
Fn308								
Fn309								

For third spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn204	MRDYC	ORCMC	SFRC	SRVC	CTH1C	CTH2C	TLMHC	TLMLC
Gn205	RCHC	RSLC	INTGC	SOCNC	MCFNC	SPSLC	*ESPC	ARSTC
Gn206	RCHHGC	MFNHGC	INCMDC	OVRC	DEFMDC	NRROC	ROTAC	INDXC
Gn207						MPOFC	SLVC	MORCMC
Gn312								
Gn313								
Gn314								
Gn315								
Fn168	ORARC	TLMC	LDT2C	LDT1C	SARC	SDTC	SSTC	ALMC
Fn169	MORA2C	MORA1C	PORA2C	SLVSC	RCFNC	RCHPC	CFINC	CHPC
Fn170							INCSTC	PC1DEC
Fn171								
Fn310								
Fn311								

Parameter

- Settings for spindle motors and spindle numbers

	#7	#6	#5	#4	#3	#2	#1	#0
3701			SS3	SS2			ISI	

[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.

#1 ISI
#4 SS2
#5 SS3 Set the number of spindles in a path.

SS3	SS2	ISI	Number of spindles in a path
"0"	"0"	"1"	0
"0"	"0"	"0"	1
"0"	"1"	"0"	2
"1"	"0"	"0"	3

NOTE
This parameter is valid when spindle serial output is enabled (bit 5 (SSN) of parameter No. 8133 is 0).

	#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 To use a serial spindle, set bit 5 (SSN) of parameter No. 8133 to 0.
2 A maximum of one analog spindle can be controlled.
3 When using an analog spindle, set it at the end of the spindle configuration.

3717	Spindle amplifier 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 spindles
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.
3: Spindle motor connected to amplifier number 3 is used.

3718	Subscript for display of a serial spindle or analog spindle

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 122
Set a subscript to be added to spindle speed display on a screen such as the position display screen.

- Connection of serial spindle control unit

	#7	#6	#5	#4	#3	#2	#1	#0
4019	PLD							

[Input type] Parameter input
[Data type] Bit spindle

#7 PLD When power is turned on, spindle amplifier parameters for the serial spindle are:
0: Not set automatically.
1: Set automatically.

When this parameter is set to 1 after the spindle motor model code parameter is set, the next power-on operation sets the standard values for the motor model in parameters and sets this parameter to 0.

4133	Spindle motor model code

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 32767
When setting initial parameters for the serial spindle, specify the spindle motor model code.

- Parameters for the serial spindle control unit

Parameter Nos. 4000 to 4539: For serial spindles
Although the above parameters are stored as CNC parameters, they are actually used by the spindle control unit of serial spindles.
For details of these parameters, refer to the following manuals of serial spindles:
FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)
FANUC SERVO MOTOR αi series MAINTENANCE MANUAL (B-65285EN)

Alarm and message

Number	Message	Description
SP1220	NO SPINDLE AMP.	Either the cable connected to a serial spindle amplifier is broken, or the serial spindle amplifier is not connected.
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 (amplifiers of even number and odd number).
SP1245 to SP1247	COMMUNICATION DATA ERROR	A communication data error was detected on the CNC. Inquire of FANUC, please.
SP1969 to SP1972	SPINDLE CONTROL ERROR	An error occurred in the spindle control software. Inquire of FANUC, please.
SP1976 to SP1979	SERIAL SPINDLE COMMUNICATION ERROR	An error occurred in the spindle control software. Inquire of FANUC, please.
SP1980 to SP1984	SERIAL SPINDLE AMP. ERROR	Defect on serial spindle amplifier. Inquire of FANUC, please.
SP1985 to SP1987	SERIAL SPINDLE CONTROL ERROR	Defective SIC–LSI on serial spindle amplifier. Inquire of FANUC, please.
SP1988, SP1989	SPINDLE CONTROL ERROR	An error occurred in the spindle control software. Inquire of FANUC, please.
SP1996	ILLEGAL SPINDLE PARAMETER SETTING	The spindle was assigned incorrectly. Check to see the following parameter. (No.3701#1,#4,#5, No.3716 or 3717)

Number	Message	Description
SP9000 or later		An alarm is issued from the spindle amplifier unit of the serial spindle. For details, refer to the manual of the serial spindle.

Diagnosis display

- Information about spindle control

	#7	#6	#5	#4	#3	#2	#1	#0
400	SIC5							

[Data type] Bit spindle

SIC5 0: The spindle interface does not operate normally.
1: The spindle interface operates normally.

403	Spindle motor temperature information

[Data type] Byte spindle

[Unit of data] °C

[Valid data range] 0to255

When the α*i* spindle interface is used, the temperature of the αi spindle motor winding is indicated. The overheat temperature varies depending on the motor.

NOTE
The overheat temperature has the following error:
Lower than 160°C: Up to 5°C
160 to 180°C: Up to 10°C

- Information about communication errors in the spindle serial output interface

	#7	#6	#5	#4	#3	#2	#1	#0
408	SSA		SCA	CME	CER	SNE	FRE	CRE

[Data type] Bit spindle

CRE 1: A CRC error occurred (warning).
FRE 1: A framing error occurred (warning).
SNE 1: The communication destination for sending/reception is incorrect.
CER 1: A reception abnormality occurred.
CME 1: No response is made during automatic scanning.
SCA 1: A communication alarm is issued in the spindle amplifier.
SSA 1: A system alarm is issued in the spindle amplifier.
(These errors are caused mainly by noise, a broken wire, a momentary power failure, and so on.)

- Indication of serial spindle load meter and speed meter

410	Spindle load meter indication (%)

411	Spindle motor speed indication (min^{-1})

[Data type] Word spindle

For correct indication of the load meter and motor speed, the following parameters must be set properly:
Maximum motor speed: Set for each axis in parameter Nos. 4020.
Load meter indication at maximum output: Set for each axis in parameter Nos. 4127.

445	Position data of position coder (pulse)

[Data type] Word spindle

NOTE
Once spindle orientation is performed after power-on, position data can be read. To read position data, set bit 1 (SPP) of parameter No. 3117 to 1.
(Except when maintenance is performed, bit 1 (SPP) of parameter No. 3117 must be set to 0.)

710	Spindle warning No.

[Data type] Byte spindle
The warning number transferred from the spindle amplifier is indicated.

712	Spindle status error No.

[Data type] Byte spindle
The status error number transferred from the spindle amplifier is indicated.

When the spindle is disabled by an error such as a PMC signal input error (for example, when two operation modes are specified at the same time), a status error is recognized while the spindle is disabled. When the signal is input normally, the normal operation is resumed.

For details of spindle error numbers, see the description of alarms (SP alarms) related to serial spindles in the alarm list.

10.4 SPINDLE ANALOG OUTPUT

Overview

Spindle analog output is a spindle motor control interface.
The spindle analog output can control up to 1 analog spindle.
This section assumes that an analog spindle is set for the first spindle (serial spindles are not used with an analog spindle, but only an analog spindle is used). When using both serial and analog spindles, see the next section, "SERIAL/ANALOG SPINDLE CONTROL".

Explanation

The terms "logical n-th spindle", "n-th spindle in a path", and "n-th spindle amplifier" used herein are defined as follows:

- Logical n-th spindle:
  The n-th spindle to be controlled as counted throughout the system
- N-th spindle (in a path):
  The n-th spindle to be controlled as counted within a path
  The terms "first spindle" and "second spindle" used herein refer to relative spindle numbers within a path. (In a single-path system, the logical n-th spindle is the n-th spindle.)

For the relationships between them, see "Spindle numbers vs. spindle motors" described later.

For how to set a spindle configuration for each path to be subjected to 2-path control, see the description of spindles in "2-path Control".

The table below lists the relationships between the spindles and functions.

Function \ Spindle	Analog spindle
	First spindle
Threading/feed per revolution (synchronous feed)	Available
Constant surface speed control	Available

Function \ Spindle	Analog spindle
	First spindle
Spindle speed fluctuation detection (T series)	Available
Actual spindle speed output (T series)	Available
Spindle positioning (T series)	Available
Cs contour control	Unavailable
Multi-spindle control	Unavailable
Rigid tapping	Available
Spindle synchronous control	Unavailable
Simple spindle synchronous control (M series)	Unavailable
Spindle control unit functions, such as spindle orientation, spindle output switching, and spindle switching (*1)	Available
Polygon machining (T series) (using the servo motor axis and spindle)	Available
Polygon machining with two spindles (T series) (Spindle and spindle polygon)	Unavailable
Spindle output control by the PMC	Available

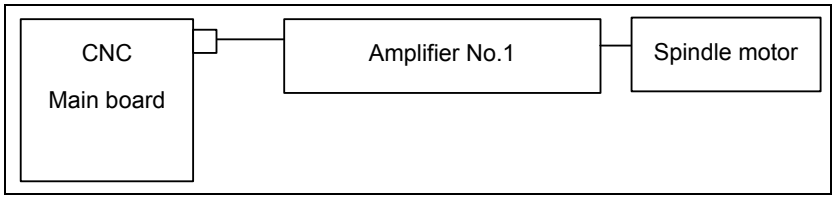
NOTE
These functions belong to the spindle control unit. They cannot be used unless the spindle control unit supports them.

- Spindle numbers vs. spindle motors

Logical spindle numbers are associated with spindle motors as follows:

1. Logical spindle numbers and amplifier numbers
   Logical spindle numbers indicate the order of spindles to be controlled (the logical arrangement order). Parameters for the individual spindles are arranged in this order. Logical spindle numbers are assigned sequentially from 1 to the maximum number of spindles.
   Amplifier numbers refer to numbers indicating the order (the order of physical placement) of spindle amplifiers connected to the CNC.
   Because only one analog spindle can be connected, the spindle amplifier and spindle motor are each assigned with number 1.

2. Setting method



Connection of analog spindle

3. Setting
   For each spindle, the type of the motor used is set in bit 0 (A/S) of parameter No. 3716, and the amplifier number, in parameter No. 3717. For spindles not to be used, the amplifier numbers are 0.

Parameter	First spindle	Second spindle
A/S(No.3716#0)	0	0
No.3717	1	0

Note

NOTE
1 If a failure occurs in the D/A converter, the following conditions occur.
- Alarm SP1241 (D/A failure) is issued.
- The spindle command voltage is set to 0.
- The spindle enable signal is set to 0.

2 The load meter value (motor load) always becomes 0 no matter what condition occurs.

Parameter

- Settings for spindle motors and spindle numbers

	#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 To use a serial spindle, set bit 5 (SSN) of parameter No. 8133 to 0.
2 A maximum of one analog spindle can be controlled.
3 When using an analog spindle, set it at the end of the spindle configuration.

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.

3718	Subscript for display of a serial spindle or analog spindle

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 122
Set a subscript to be added to spindle speed display on a screen such as the position display screen.

	#7	#6	#5	#4	#3	#2	#1	#0
3798								ALM

[Input type] Parameter input
[Data type] Bit

#0 ALM The spindle alarm (SP****) for all spindles is:
0: Enabled.
1: Ignored.
When this parameter is set to 1, the spindle-related alarms are ignored. So, be sure to set this parameter to 0 at all times except for special cases such as maintenance.

	#7	#6	#5	#4	#3	#2	#1	#0
3799							NDPs	NALs

[Input type] Parameter input
[Data type] Bit spindle

#0 NALs An alarm detected on the spindle amplifier side is:
0: Displayed.
1: Not displayed.
(This parameter is valid when bit 0 (ALM) of parameter No. 3798 is set to 0.)
When this parameter is set to 1, an alarm detected on the spindle amplifier side is ignored. So, be sure to set this parameter to 0 at all times except for special cases such as maintenance.

#1 NDPs When an analog spindle is used, a position coder disconnection check is:
0: Made.
1: Not made.
(This parameter is valid when bit 0 (NAL) of parameter No. 3799 is set to 0.)

When no position coder is used with an analog spindle, set this parameter to 1.

NOTE
When an analog spindle is used at the same time under serial/analog spindle control, a position coder cannot be used for the analog spindle.
Be sure to set this parameter to 1.

Alarm and message

Number	Message	Description
SP1241	D/A CONVERTER ERROR	The D/A converter for controlling analog spindles is erroneous.
SP1996	ILLEGAL SPINDLE PARAMETER SETTING	The spindle was assigned incorrectly. Check to see the following parameter. (No.3701#1,#4,#5, No.3716 or 3717)

10.5 SERIAL/ANALOG SPINDLE CONTROL

Overview

Serial/analog spindle control allows an analog spindle to be used as one of the controlled spindles in the entire system. Only spindle speed command control and spindle speed command control by the PMC can be used.

NOTE
1 When using a serial spindle, set bit 5 (SSN) of parameter No. 8133 to 0.
2 The analog spindle used must be set to the last of the spindle configuration.

Explanation

Serial/analog spindle control allows only one analog spindle to be used as one of the controlled spindles in the entire system. The analog spindle used must be set to the last spindle of the spindle configuration.

Spindle functions available in serial/analog spindle control

The table below lists the relationship between the spindles and functions:

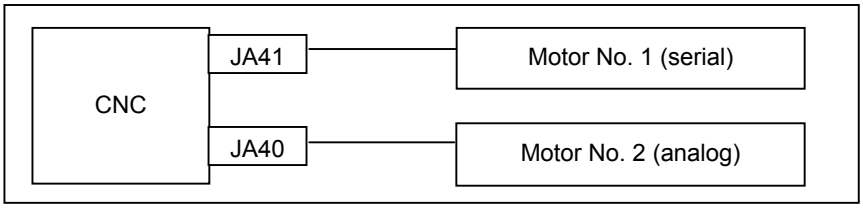
Function \ Spindle	Serial spindle (*1)	Analog spindle
Threading/feed per revolution (synchronous feed)	Available	Unavailable
Constant surface speed control	Available	Unavailable
Spindle speed fluctuation detection	Available	Unavailable
Actual spindle speed output (T series)	Available	Unavailable
Spindle positioning (T series)	Available	Unavailable
Cs contour control	Available	Unavailable
Rigid tapping	Available	Unavailable
Spindle synchronous control	Available	Unavailable
Simple spindle synchronous control (M series)	Available	Unavailable
Spindle control unit functions, such as spindle orientation, spindle output switching, and spindle switching (*2)	Available	Available
Polygon machining (T series) (using the servo motor axis and spindle)	Available	Unavailable
Polygon machining with two spindles (T series) (Spindle and spindle polygon)	Available	Unavailable
Spindle output control by the PMC	Available	Available

NOTE
1 When using a serial spindle, set bit 5 (SSN) of parameter No. 8133 to 0.
2 These functions belong to the spindle control unit. They cannot be used unless the spindle control unit supports these functions.

Connection example

Connect the analog spindle to JA40.
When an analog spindle is used under serial/analog spindle control, a position coder cannot be attached to the analog spindle.
For connection of serial spindles, refer to "Connection Manual (Hardware)" (B64303EN).

- **When connecting one serial spindle and one analog spindle (when the maximum number of controllable spindles is two)**



Restrictions

When an analog spindle is used under serial/analog spindle control, there are restrictions as follows:

1 Be sure to set the analog spindle as the last spindle of the controlled spindles.
Example:
When there are two controlled spindles, and one of these spindles is an analog spindle
The controlled spindle number of the analog spindle is 2.
Set the spindle amplifier number of the analog spindle (parameter No. 3717) to 2.
2 A position coder cannot be installed.
3 Only spindle speed command control and spindle speed command control by the PMC can be used. Other control and spindle functions cannot be used.

Parameter

	#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.

NOTE
Make the setting on the side of the path in which multi-spindle control is unnecessary in 2-path control.

	#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 To use a serial spindle, set bit 5 (SSN) of parameter No. 8133 to 0.

NOTE
2 A maximum of one analog spindle can be controlled.
3 When using an analog spindle, set it at the end of the spindle configuration.

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 3 is used.
3: Spindle motor connected to amplifier number 3 is used.

NOTE
When using an analog spindle, set it at the end of the spindle configuration.
(Example)
When there are three spindles in an entire system (two serial spindles and one analog spindle), set the spindle amplifier number (this parameter) of the analog spindle to 3.

3718	Subscript for display of a serial spindle or analog spindle

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 122
Set a subscript to be added to spindle speed display on a screen such as the position display screen.

	#7	#6	#5	#4	#3	#2	#1	#0
3798								ALM

[Input type] Parameter input
[Data type] Bit

#0 ALM The spindle alarm (SP****) for all spindles is:
0: Enabled.
1: Ignored.
When this parameter is set to 1, the spindle-related alarms are ignored. So, be sure to set this parameter to 0 at all times except for special cases such as maintenance.

	#7	#6	#5	#4	#3	#2	#1	#0
3799							NDPs	NALs

[Input type] Parameter input
[Data type] Bit spindle

#0 NALs An alarm detected on the spindle amplifier side is:
0: Displayed.
1: Not displayed.
(This parameter is valid when bit 0 (ALM) of parameter No. 3798 is set to 0.)
When this parameter is set to 1, an alarm detected on the spindle amplifier side is ignored. So, be sure to set this parameter to 0 at all times except for special cases such as maintenance.

#1 NDPs When an analog spindle is used, a position coder disconnection check is:
0: Made.
1: Not made.
(This parameter is valid when bit 0 (NAL) of parameter No. 3799 is set to 0.)
When no position coder is used with an analog spindle, set this parameter to 1.

NOTE
When an analog spindle is used at the same time in serial/analog spindle control, a position coder cannot be used as the analog spindle. Be sure to set this parameter to 1.

10.6 SPINDLE SPEED CONTROL

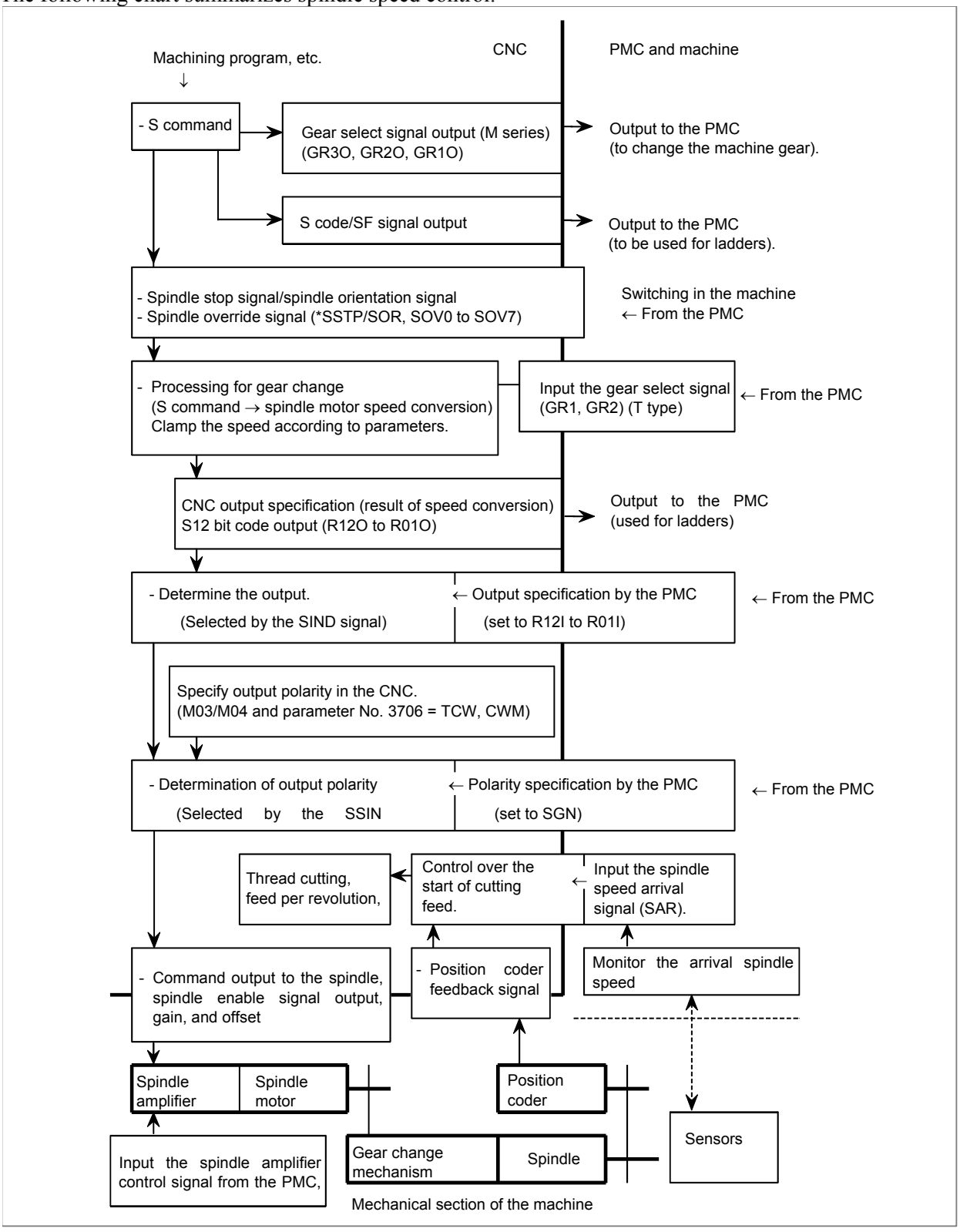
Explanation

This section explains the following in relation to spindle speed control:

- Command flow in spindle speed control
- Processing for gear change (M and T type gear selection methods)
- Position coder feedback signal
- Speed arrival signal (SAR)

- Command flow of spindle speed control

The following chart summarizes spindle speed control.



- S command

The S command specifies the spindle speed entered from machining programs, etc. for the CNC.
For constant surface speed control (during G96 mode), the CNC converts the specified surface speed to the spindle speed.

M

In the M series, when constant surface speed control is not provided (bit 0 (SSC) of parameter No. 8133 is set to 0) and bit 4 (GTT) of parameter No. 3706 is set to 0, the CNC determines the necessary gear stage for obtaining a certain spindle speed, based on the settings of parameter Nos. 3741, 3742, 3743, and so on and the specified S value, and notifies the PMC of the gear stage.
(GR3O, GR2O, GR1O<Fn034.2, 1, 0>)

- S code/SF signal output

When spindle serial output is enabled (bit 5 (SSN) of parameter No. 8133 is set to 0), the CNC's spindle control functions, internally converting a specified S value into an output to the spindle motor. Furthermore, the response with the S code/SF signal to an S command input differs from when neither gear changing nor the constant surface speed control function (bit 0 (SSC) of parameter No. 8133 is set to 0) is enabled, as follows (see below).

M

M series → Outputs the S code.
The SF signal is output only when the CNC directs the PMC to change the gear.

T

T series → Outputs neither S code nor SF signal.
(When constant surface speed control is provided (bit 0 (SSC) of parameter No. 8133 is set to 1), it is not certain that the S command does not always indicates the spindle speed.)

If you use the S code and SF signal for processing in the PMC ladder, you must specify parameters related to parameter No. 3705.

- Spindle stop signal(*SSTP)

When the CNC performs spindle speed control, the speed command output to the spindle is set to 0 by this signal. Setting this signal to 0 stops the spindle.
To enable speed commands to be issued to the spindle, set this signal to 1.

- Spindle orientation signal (SOR)

If the spindle orientation signal is logical 1 and the spindle stop signal is logical 0, the spindle rotates in the direction specified by bit 5 (ORM) of parameter No. 3706 at a constant speed specified by parameter No. 3732.
Because the spindle rotates at a constant speed regardless of the gear stage, this signal can be used to rotate the spindle to drive the stopper or pin during mechanical spindle positioning.

M

For the M series, setting parameter GST (bit 1 of parameter No. 3705) enables the spindle motor to rotate at a constant speed. This function can be used for gear shifting because it maintains a constant speed of the gear change mechanism.

- Spindle speed override signal (SOV00 to SOV07)

This signal specifies an override of 0% to 254% for the specified S value for spindle control.
However, the spindle speed override function is disabled when the CNC is in the following state:
Tapping cycle (M series : G84, G74 T series : G84, G88)
Thread cutting (M series : G33 T series : G32, G92, and G76)
When the spindle speed control is performed but the spindle speed override is not used, set the override value to 100%.

- Processing for gear changing

Although the S command contains the spindle speed, the object that is actually controlled is the spindle motor. Therefore, the CNC must have some provision to detect the gear stage between the speed and spindle motor.
There are two types of gear selection methods:
M type (for the M series only)
The CNC selects a gear stage according to the range of speed for each gear stage previously specified in a parameter, as directed by the S command, and informs the PMC of the selected gear stage (one of the three gear stages) using the gear select signal output (GR3O, GR2O, GR1O).
Also, the CNC outputs the spindle speed based on the selected gear stage (output as the gear select signal).
T type
The gear stage to be used is determined on the machine side, and the gear stage is input using the gear selection signals (GR1 and GR2) (one of the four gear stages).
The CNC outputs the appropriate speed command for the selected gear range.

- Selection of gear change system

M

The M series can use either M or T type.
M type → Constant surface speed control is not provided (bit 0 (SSC) of parameter No. 8133 is set to 0), and bit 4 (GTT) of parameter No. 3706 is set to 0.
T type → Constant surface speed control is provided (bit 0 (SSC) of parameter No. 8133 is set to 1), and bit 4 (GTT) of parameter No. 3706 is set to 1.

NOTE
The M type gear selection method cannot be selected when one of the following functions is used:
- Constant surface speed control
- Multi-spindle control

T

The T series can use only T type.

- M type gear selection method

By specifying from S0 to S99999 in memory or MDI operation, the CNC outputs a command corresponding to the spindle speed. There is a two-speed (GR1O and GR2O) or three-speed range (GR1O, GR2O, GR3O), set by parameter Nos. 3741-3743, and the gear selection signal is output simultaneously.
When the gear selection signal is changed, the SF signal is output at the same time.
By setting bit 6 (SFA) of parameter No. 3705, SF can also be output regardless of gear change.
The meaning of the gear signals is shown below:

	Gear 2-stage	Gear 3-state	Remarks
GR1O	Low	Low	Low :Low Gear
GR2O	High	Middle	Middle :Middle Gear
GR3O		High	High :High Gear

The speed commands output to the spindle motor are as follows:
- For the serial spindle, the speed commands are processed as values 0 to 16383 between the CNC and spindle control unit.
- For the analog spindle, the speed commands are output to the analog voltage signal SVC as analog voltages 0 to 10 V.

The following descriptions exemplify the analog spindle. However, they can be applied to the serial spindle on the assumption that spindle motor speed with analog voltage 10 V corresponds to the maximum spindle motor speed.

(i) M type gear change method A (See Fig. 10.6 (a).)

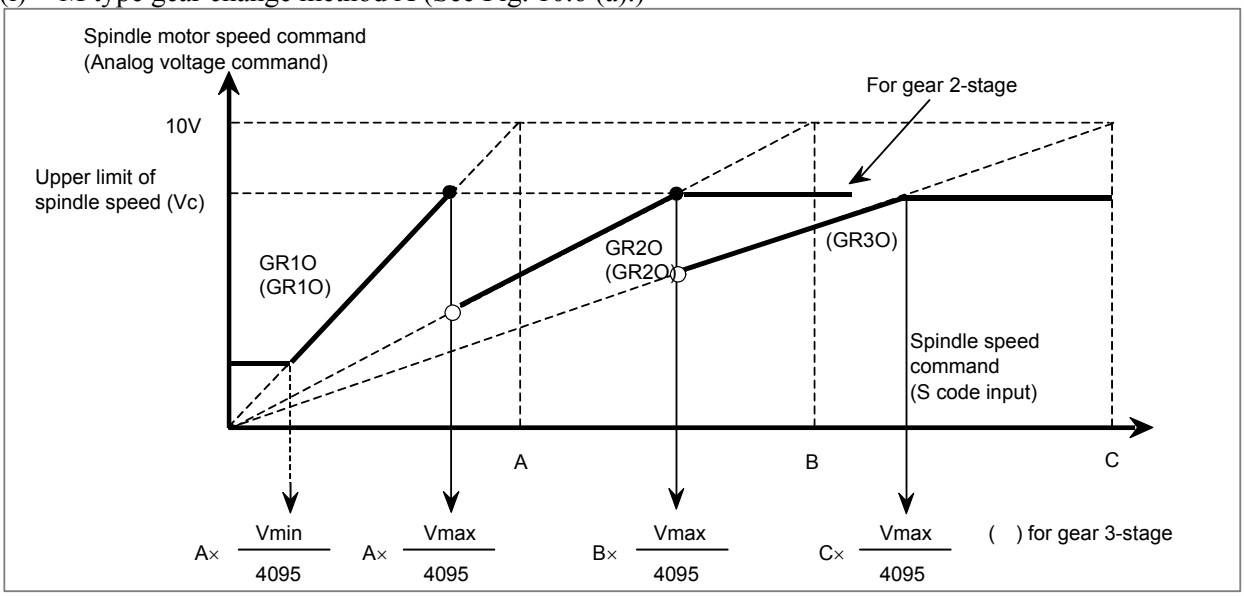


Fig. 10.6 (a) S code input and output voltage

Set the following values as parameters:

- Constant Vmax: for upper limit of spindle speed (parameter No. 3736)

$$Vmax = 4095 \times \frac{\text{Upper limit of spindle speed}}{\text{Spindle speed at command voltage 10V}}$$

- Constant Vmin; for lower limit of spindle speed (parameter No. 3735)

$$Vmin = 4095 \times \frac{\text{Lower limit of spindle speed}}{\text{Spindle speed at command voltage 10V}}$$

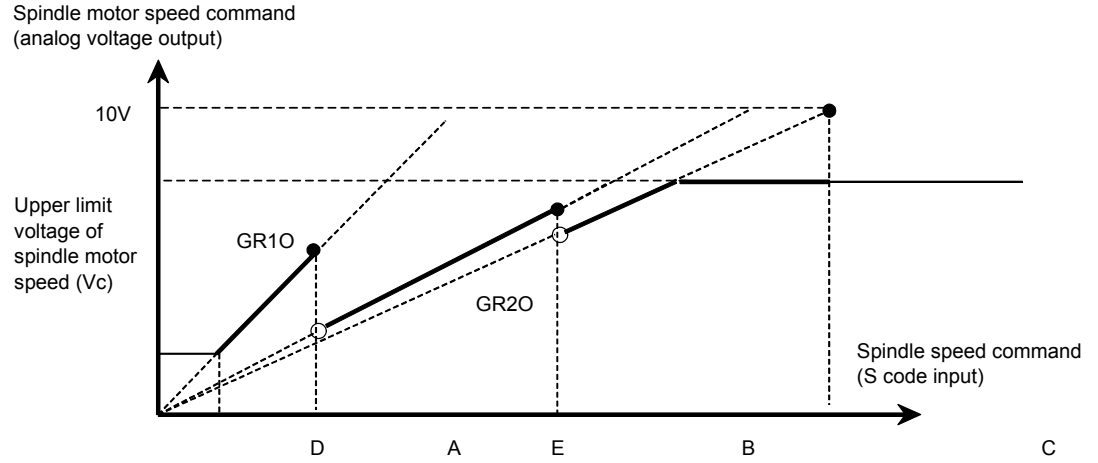
- Spindle speed A (min^{-1}) : at command voltage 10V and low gear (parameter no. 3741)
- Spindle speed B (min^{-1}) : at command voltage 10V and high gear (or middle-high gear) (parameter no. 3742)
- Spindle speed C (min^{-1}) : at command voltage 10V and high gear (parameter no. 3743)

NOTE

If a specified voltage of 10 V is already higher than the acceptable input voltage for the spindle drive system, calculate hypothetically the spindle speed that corresponds to 10 V using a proportional calculation method and use it instead. Now, in response to the specified S code, the speed command and gear select commands (GR3O, GR2O, GR1O) are output to the spindle motor as shown in Fig. 10.6 (a)

(ii) Gear change point during tapping cycle mode (G84, G74)

In case of G84 (tapping cycle) or G74 (counter tapping cycle) the gear shift speed is changed by parameter SGT(No. 3705#3). In this case, gear shift is performed at the speed set by parameter Nos. 3761 and 3762 (Fig. 10.6 (b)).

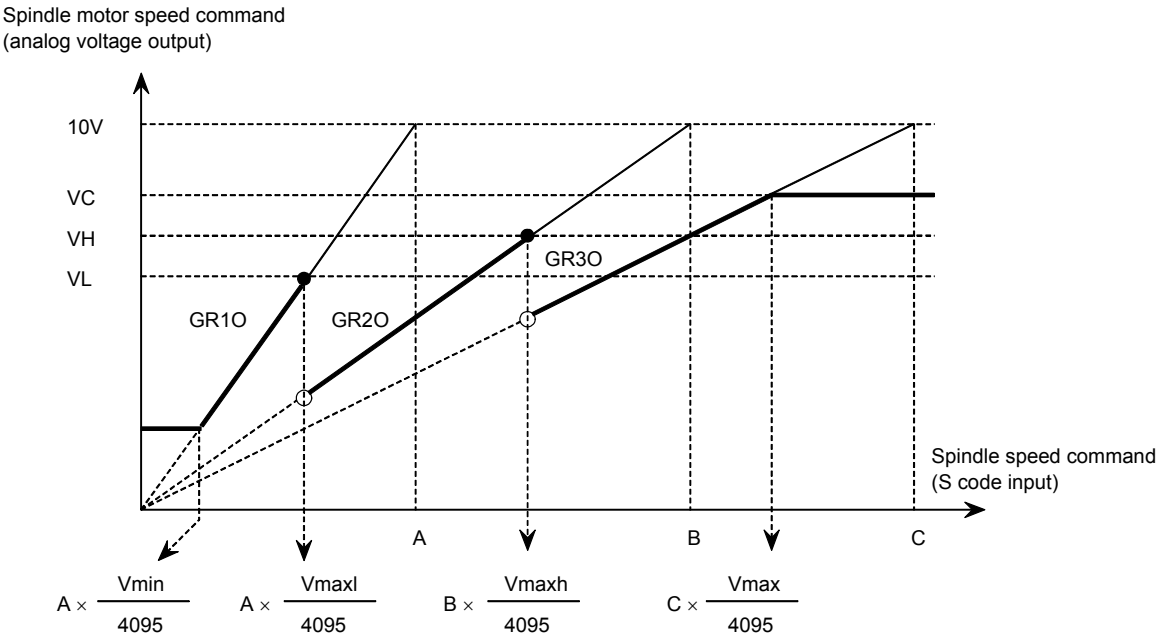


D : Gear shift point (parameter No. 3761) for low/high speed change (or low/medium speed change) (set by parameter as min^{-1} value) during tapping.
E : Gear shift point (parameter No. 3762) (3-stage gear) for medium/high speed change (set by parameter as min^{-1} value) during tapping.

Fig. 10.6 (b) S code input and output voltage (in tapping)

M

(iii) M type gear change method B (M series) (See Fig. 10.6 (c).)
The speed (min^{-1}) at which the low-speed and the high-speed gears are changed can be set as a parameter (Nos.3751 and 3752) by setting parameter SGB (No. 3705# 2). When a 3-stage gear is used, it is possible to set the speeds (min^{-1}) for switching low-speed and medium-speed gears, and medium-speed and high-speed gears, using parameters Nos. 3751 and 3752.



VC: Voltage corresponding to the upper limit of output value to spindle motor.
VH: Voltage corresponding to the upper limit of output value to spindle motor with high-speed gears (medium speed gear for 3-stage gear)
VL: Voltage corresponding to the upper limit of output value to spindle motor at low-speed gear

Fig. 10.6 (c) M type gear change B

When using this function, set the following parameters:

- Constant Vmax (Parameter No.3736) related to the upper limit of spindle motor speed (min^{-1})

$$Vmax = 4095 \times \frac{\text{Upper limit of spindle motor speed}}{\text{Spindle motor speed when the command voltage is 10V}}$$

- Constant Vmin (Parameter No. 3735) related to the lower limit of spindle motor speed (min^{-1})

$$Vmin = 4095 \times \frac{\text{Lower limit of spindle motor speed}}{\text{Spindle motor speed when the command voltage is 10V}}$$

- Constant Vmaxl (Parameter No. 3751) related to the upper limit of spindle motor speed (min^{-1}) with low-speed gears

$$Vmaxl = 4095 \times \frac{\text{Upper limit of spindle motor speed with low-speed gears}}{\text{Spindle motor speed when the command voltage is 10V}}$$

- Constant Vmaxh (Parameter No. 3752) related to the upper limit of spindle motor speed (min^{-1}) with high-speed gears (medium-speed gear for 3-stage gear)

$$Vmaxh = 4095 \times \frac{\text{Upper limit of spindle motor speed with high-speed gears}}{\text{Spindle motor speed when the command voltage is 10V}}$$

- Spindle speed A (Parameter No.3741) (min^{-1}) with low-speed gears when the command voltage is 10V
- Spindle speed B (Parameter No.3742) (min^{-1}) with high-speed gears when the command voltage is 10V (medium-speed gear for 3-stage)
- Spindle speed C (Parameter No.3743) (min^{-1}) with high-speed gears when the command voltage is 10V (3-stage gear)

Spindle motor speed commands (0 to 10V) and gear selecting signals (GR1O, GR2O, GR3O) are issued on each S code command as shown in the Fig. 10.6 (c):

⚠ CAUTION

1 In a tapping cycle when parameter SGT (No. 3705 #3) is set, the gears are changed over at the gear changing point for tapping.
2 For this function (parameter SGB (No. 3705#2) = 1), when only one-stage gear is used, the voltage corresponding to the upper limit value to the spindle motor is calculated using Vmaxl, and when 2-stage gear is used, it is calculated according to Vmaxh. Therefore, when SGB is 1, set Vmaxl when only one-stage gear is used, Vmaxl and Vmaxh when 2-stage gear is used.

Bit 6 (GMB) of parameter No. 3712 can be used to select the maximum spindle motor speed for each gear.

When the bit 6 (GMB) of parameter No. 3712 is set to 0:

The maximum motor speed when each gear is used is clamped by parameters No. 3741 to 3743 or parameter No. 3736.

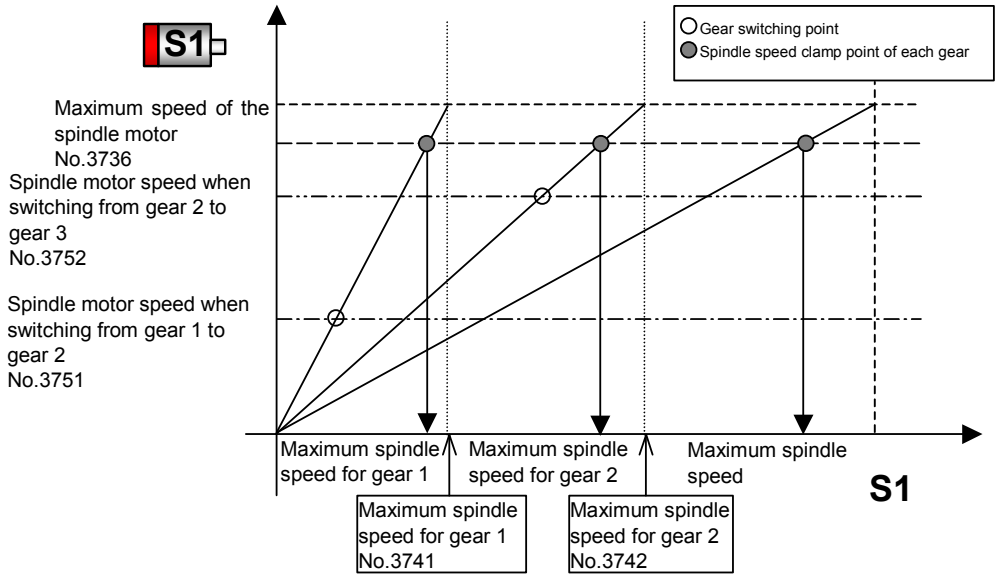


Fig. 10.6 (d) When the bit 6 (GMB) of parameter No. 3712 is set to 0

When the bit 6 (GMB) of parameter No. 3712 is set to 1:

The maximum motor speed of gear 1 is clamped at the motor speed of the switching point from gear 1 to gear 2 and the maximum motor speed of gear 2 is clamped at the motor speed of the switching point from gear 2 to gear 3.

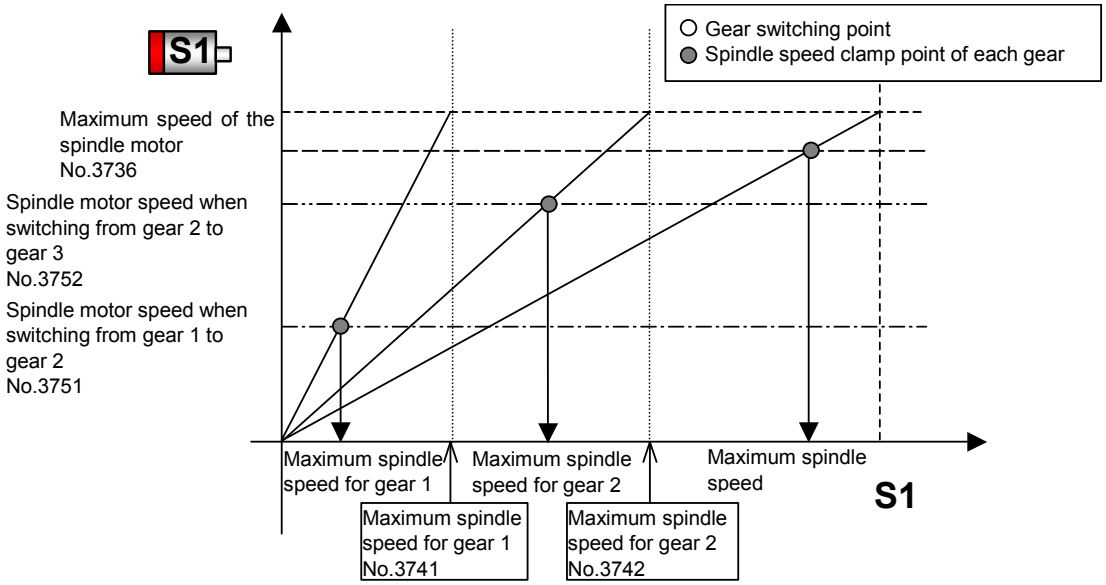
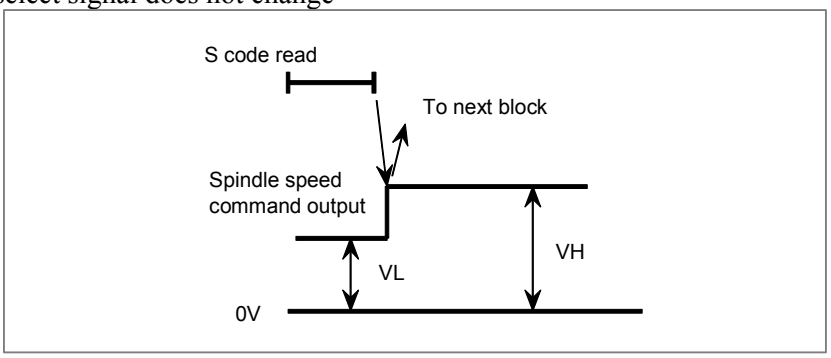


Fig. 10.6 (e) When the bit 6 (GMB) of parameter No. 3712 is set to 1

- Time chart

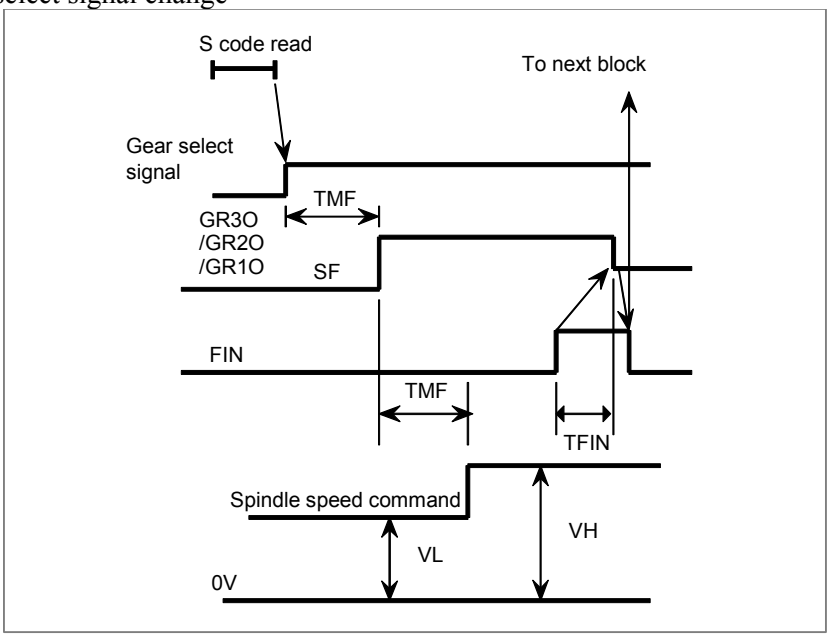
When S code is commanded, the I/O signal time chart is :

(i) When Gear select signal does not change



In this case, the SF signal is not output and the CNC advances to the next block automatically after the next spindle speed command is output.

(ii) When Gear select signal change



In this case, the gear select signal is output; after elapse of the time constant set by parameter (TMF), the SF signal is output. After another TMF elapse, the spindle speed command is output. On the PMC side, change the gears by this SF signal, and return the FIN signal after the end of gear change. The time chart for SF and FIN signals is the same as in S code output. TMF, set by parameter No. 3010, is common to M, S and T functions.

Moreover, specifying bit 6 (SFA) of parameter No. 3705 can specify that the SF signal be output even if no gear change is used.

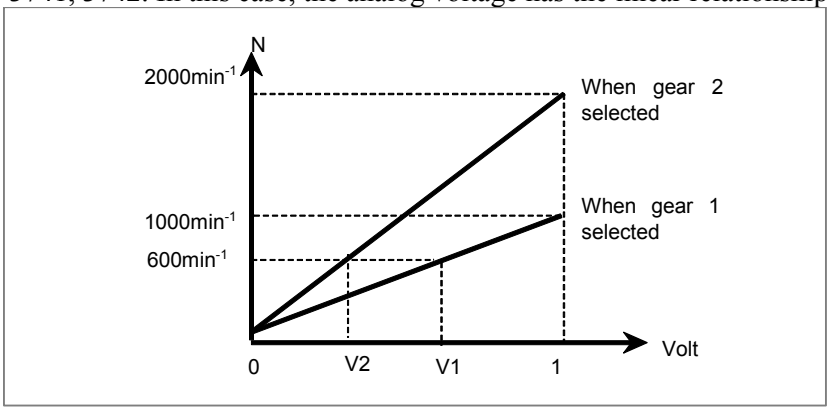
- Details of T type (Input of GR1, GR2)

To perform the T type gear change, the maximum spindle speed for each gear side must be set in parameters Nos. 3741 to 3744.

The gear select signal is a 2 bit code signal (GR1, GR2). The relationship between the signal and gear number is :

Gear select signal		Gear No.	Parameter No. for max. spindle speed
GR1	GR2		
0	0	1	No. 3741
1	0	2	No. 3742
0	1	3	No. 3743
1	1	4	No. 3744

The following descriptions apply to the analog spindle. Like the descriptions of the M type, they also apply to the serial spindle on the assumption that spindle motor speed with analog voltage 10 V corresponds to the maximum spindle motor speed.
In addition, for the speed command output to the spindle motor, analog voltages 0 to 10 V for analog spindle control correspond to digital data 0 to 16383 for serial spindle control. However, it might be easier if you consider them code signals from 0 to 4095 for convenience sake without distinguishing between serial and analog spindles.
Assume that gear switching is two stage switching. If the spindle speed with the output voltage 10 V is 1000 min^{-1} for the low speed gear (G1) and 2000 min^{-1} for the high speed gear (G2), set these speeds by the parameter No. 3741, 3742. In this case, the analog voltage has the linear relationship shown below.



When spindle speed S=600 is given, V1 (for G1) or V2 (for G2) is calculated inside the CNC and output to the machine side.
V1 ; 6 (V)
V2 ; 3 (V)
The value of output voltage V is calculated automatically from the following equations:
V = 10N / R
R : Spindle speed at 10V output voltage
N : Spindle speed given by S command of 5 digits
This is equivalent to the G97 mode for constant surface speed control.
See "CONSTATNT SURFACE SPEED CONTROL" for operations during the constant surface speed control mode (G96).

- Determining spindle speed command output R12O-R01O (Output) R12I-R01I (Input) SIND (Input)

Using the above processing for gear change, the CNC calculates the speed command output to the spindle motor that is necessary to obtain the specified spindle speed with the gear.
The calculation result is output to the PMC through S12-bit code signals indicating a value 0 to 4095.
(R12O to R01O<F037.3 to F036.0>)
Then, according to the selection by the SIND signal <Gn033.7>, the speed command output to the spindle motor is determined.

SIND signal <Gn033.7>	Speed command output to spindle motor
0	Speed command output calculated by CNC
1	Speed command output set by PMC Also see "SPINDLE OUTPUT CONTROL BY THE PMC".

- Polarity of spindle speed command output

The speed command output to the spindle motor is determined as described above, but the actual output polarity is determined by the CNC as follows:

Parameter TCW (No.3706#7)	Parameter CWM (No.3706#6)	Output polarity
0	0	+(M03/M04 command)

Parameter TCW (No.3706#7)	Parameter CWM (No.3706#6)	Output polarity
0	1	-(M03/M04 command)
1	0	+(M03 command), -(M04 command)
1	1	-(M03 command), +(M04 command)

Then, according to the selection by the SSIN signal <Gn033.6>, the output polarity of the speed command output to the spindle motor is determined.

SSIN signal <Gn033.6>	Output polarity of speed command output to spindle motor
0	Output polarity determined by CNC
1	Output polarity set by PMC Also see "SPINDLE OUTPUT CONTROL BY THE PMC".

NOTE
When bit 7 (TCW) of parameter No. 3706 is 1, and the output polarity is to be determined by the CNC, the CNC cannot determine the output polarity if M03 or M04 has not been specified even once since power-on. As a result, no command is output to the spindle.

- Command output to the spindle control unit

According to the speed command output and the polarity determined so far, the command is sent to the spindle control unit as follows:

- For serial spindle → Digital data 0 to ±16383
- For analog spindle → Analog voltage 0 to ±10 V

- Conditions for stopping spindle speed command output

A command being output to the spindle is set to 0 when *SSTP is set to 0 and when a command that sets the spindle speed command output to 0 such as a S0 command is specified.
With M05, emergency stop, and a reset, the CNC does not set the command output to the spindle to 0.

- Position coder feedback signal

The position coder is necessary for thread cutting or feed per revolution.
The position coder detects the actual spindle speed and the one-rotation signal (used to detect a fixed point on the spindle for threading). Any number of pulses output from the position coder can be selected, and it is set in parameter No. 3720.
When the gear ratio between the position coder and spindle is to be used, set the gear ratio for the position coder and spindle in parameter Nos. 3721 and 3722, respectively.
See "RIGID TAPPING" for position coder connection for rigid tapping.

- Speed arrival signal (SAR)

The spindle speed arrival signal SAR is an input signal used as a condition for the CNC to start cutting feed. This signal is used generally when cutting feed should be started after the spindle reaches the specified speed.
In this case, a sensor is used to check the spindle speed. The detected speed is sent to the CNC via the PMC.
When the above operation is performed continuously using the PMC ladder, however, cutting feed may be started based on the SAR signal indicating the previous spindle state (spindle speed before change), if the spindle speed change command and the cutting feed command are issued at the same time.
To avoid the above problem, monitoring the SAR signal can be deferred for a time specified by parameter No. 3740 after the S command or cutting feed command was issued.
When using the SAR signal, set bit 0 (SAR) of parameter No. 3708 to 1.

Item No. 0000 (SPEED ARRIVAL ON) on the diagnosis screen is kept at 1 while this function is keeping the cutting feed block at a halt.

- Analog spindle

Gain and offset

The analog spindle may require gain and offset voltage adjustment depending on the spindle motor speed amplifier being used.
The following parameters are available for such adjustment.

Gain adjustment data: Parameter No. 3730
Offset voltage compensation: Parameter No. 3731

Spindle enable signal (ENB)

Another output related to spindle control is the spindle enable signal ENB.
The ENB signal is logical 1 when a nonzero command output is sent to the spindle. If the command is logical 0, the ENB signal becomes logical 0.
When the analog spindle is being used, an offset voltage in the spindle motor speed amplifier may cause the spindle motor to rotate at low speed even if the command output (in this case, analog voltage) to the spindle is zero. The ENB signal can be used to stop the motor in such a case.

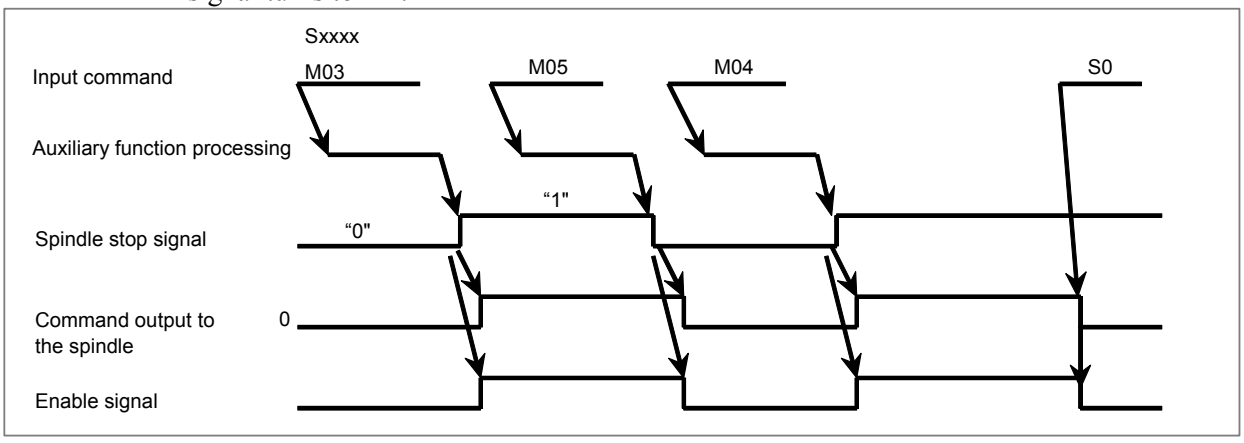
Electrical specification of analog spindle interface

The ENB1/2 signal on the analog spindle interface is an interface signal that is turned on and off under the same conditions as of the spindle enable signal <Fn001.4>, and it can be used even when a serial spindle is used.
For related signals on the analog spindle interface, refer to the section of the analog spindle in "Connection Manual (Hardware)".

Signal

Spindle stop signal *SSTP<Gn029.6>

[Classification] Input signal
[Function] The command output to the spindle is disabled.
[Operation] When the spindle stop signal turns to "0" , the output voltage becomes 0V and the enable signal ENB turns to "0" (M05 is not output).
When this signal turns to "1", the analog voltage returns to its original value and the ENB signal turns to "1".



The above time chart is an example. Actually, the time chart should meet the specification of the spindle control unit.

- When this signal is not used, always set the signal to "1".
- When the M03, M04, or M05 command is issued, the CNC outputs the code signal and strobe signal only.

Spindle orientation signal SOR<Gn029.5>

[Classification] Input signal

[Function] The spindle or the spindle motor is rotated at a constant

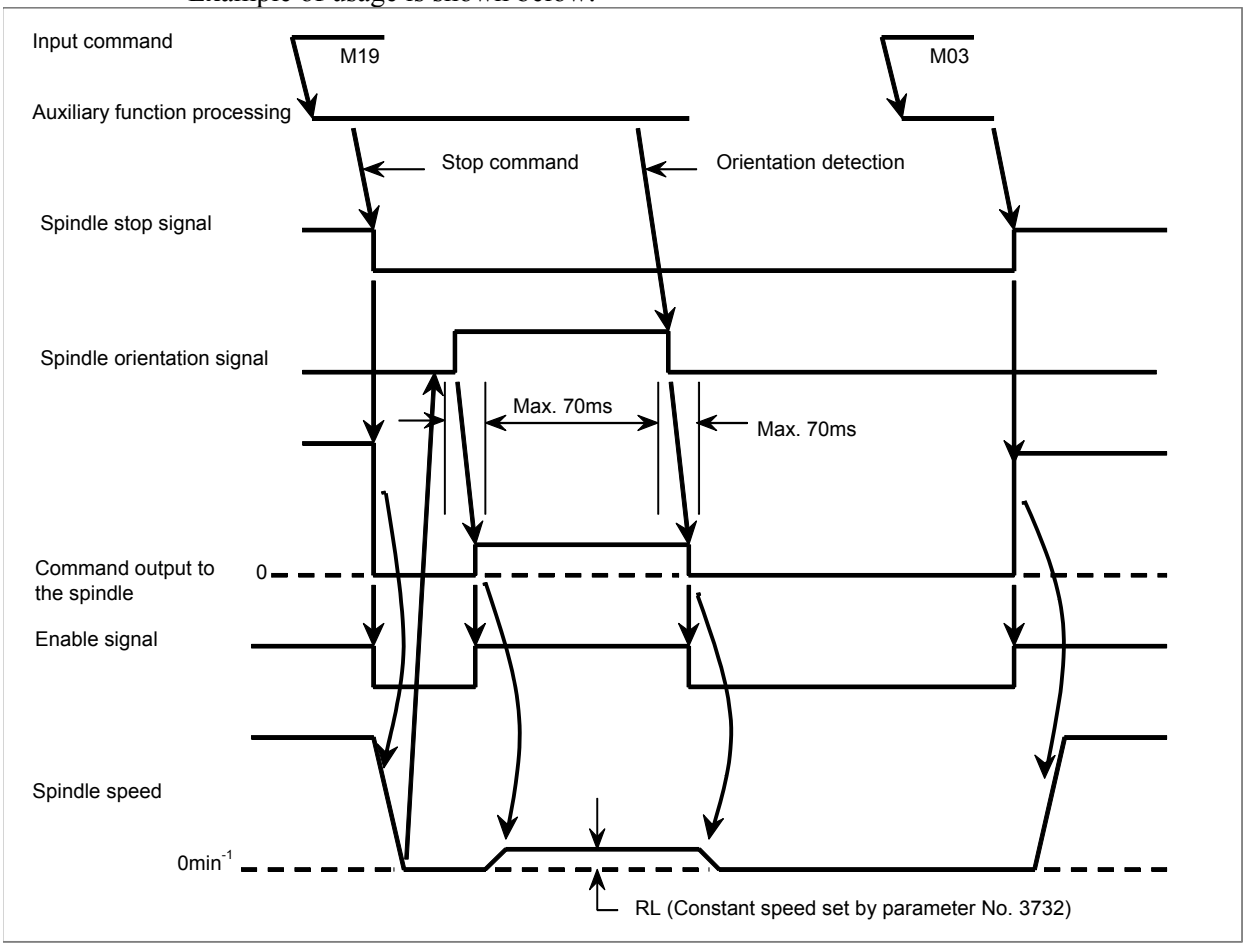
[Operation] When the spindle orientation signal turns to "1" and the spindle stop signal *SSTP turns to "0", a spindle speed command which lets the spindle rotate at the constant speed set by parameter No. 3732 is output. The enable signal ENB also turns to "1". This signal is disabled when the spindle stop signal is "1".

In the machining center system, when the spindle speed for orientation is set by parameter GST No. 3705#1 and the SOR signal is input, the CNC outputs the spindle speed command corresponding to the speed set to parameter 3732 with an output polarity set by parameter ORM (No. 3706#5), but the gear select signal does not change.

For example, if the SOR signal is turned to "1" with high gear selected, and the speed set to parameter No. 3732 is in the low gear range, the gear select signal does not change and the command output is calculated and output to obtain the set speed at high gear.

When the spindle motor speed is set by parameter GST (No. 3705#1)=1, the command output is output regardless of gear select signal. When the spindle motor speed is set, it is used for gear shift.

Example of usage is shown below:



Spindle speed override signals SOV0 to SOV7<Gn030>

[Classification] Input signal

[Function] The spindle speed override signal specifies an override from 0% to 254% in 1% units for the S command sent to the CNC.

[Operation] An override value in binary must be set in 8 bits from SOV7 to SOV0.

The spindle speed override function is disabled (an override of 100% is applied) under the following conditions:

- Tapping cycle (M series : G84, G74 T series : G84, G88)

- Thread cutting (M series : G33 T series : G32, G92, G76)

→ When this function is not in use, specify an override of 100%; otherwise, an override of 0% becomes effective, thus disabling the spindle from rotating.

Spindle speed arrival signal SAR<Gn029.4>

[Classification] Input signal

[Function] The SAR signal initiates cutting feed. In other words, if the signal is logical 0, cutting feed will not start.

[Operation] Generally, this signal is used to inform the CNC that the spindle has reached the specified speed.

For this purpose, the signal must be set to 1 only after the actual speed of the spindle has reached the specified speed.

Setting parameter No. 3740 with a wait time before the start of checking the SAR signal inhibits cutting feed from starting under a condition of SAR = 1 specified before the change of the spindle command.

To use the SAR signal, it is necessary to set bit 0 (SAR) of parameter No. 3708 to 1.

The CNC checks the SAR signal under the following conditions:

a. Bit 0 (SAR) of parameter No. 3708 is set to 1.
b. Before starting distribution of the first feed (move command) block after shifting from the rapid traverse mode to the cutting feed mode. This checking is performed after the time set by parameter No. 3740 has elapsed after the feed block is read.
c. Before starting distribution of the first feed command block after an S code is commanded. The wait time for checking is the same as in item b.
d. When an S code and feed are programmed in the same block, the S code (or command output to the spindle) is output, and the SAR signal is checked after a fixed time elapses. If the SAR signal is set to "1", feed begins.

> ⚠ **CAUTION**
> According to the conditions of item d above, note that if the circuit is so designed that SAR is turned to "0" simultaneously with the output of an S code and the change of spindle speed is initiated by the DEN signal, the operation will stop. That is, the spindle speed does not reach the commanded speed because the CNC is waiting for the DEN signal and distribution is not started because the CNC is waiting for the SAR signal.

Spindle enable signal ENB<Fn001.4>

[Classification] Output signal

[Function] Informs absence or presence of spindle output command.

[Output cond.] The ENB signal becomes logical 0 when the command output to the spindle becomes logical 0. Otherwise, the signal is logical 1.

During analog spindle control, S0 may not be able to stop the spindle from rotating at low speed because of an offset voltage in the spindle motor speed control amplifier. In such a case, the ENB signal can be used to provide a condition to determine whether to stop the motor.

The analog spindle interface has electric signals (ENB1 and ENB2) similar to the ENB. These signals work under the same conditions as with the ENB signal.

The ENB signal can be used also for serial spindle control.

Gear selection signals GR1O,GR2O,GR3O<Fn034.0 to .2>

[Classification] Output signal

[Function] The gear select signal specifies a gear stage to the PMC.

[Output cond.] For details of this signal, see descriptions on the M type gear selection method in explanation of this section.

Gear selection signals GR1,GR2<Gn028.1, .2>

[Classification] Input signal

[Function] This signal informs the CNC of the gear stage currently selected.

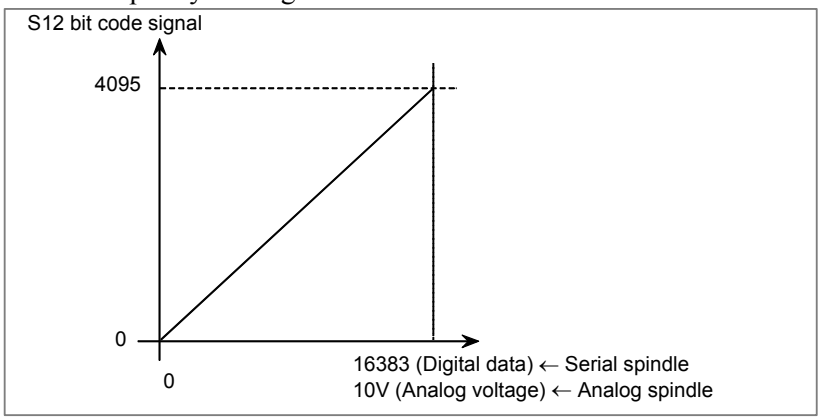
[Operation] For details of this signal, see descriptions on the T type gear selection method in explanation of this section.

S12-bit code signals R01O to R12O<Fn036.0 to Fn037.3>

[Classification] Output signal

[Function] These signals convert the spindle speed command value calculated by the CNC to code signals 0 to 4095.

[Output cond.] The relationship between the spindle speed command value (calculated by the CNC) and the value output by this signal is as shown below.



These signals convert the spindle speed command value calculated by the spindle control function of the CNC to data from 0 to 4095 (for both serial and analog spindle control) and outputs the result. This does not match the command set by spindle output control of the PMC. (See "SPINDLE OUTPUT CONTROL BY THE PMC".)

- Other signals

Spindle speed function code signals S00 to S31<Fn022 to Fn025> (Output)
Spindle speed function strobe signal SF<Fn007.2> (Output)

See "SPINDLE SPEED FUNCTIONS" and "SPINDLE OUTPUT CONTROL BY THE PMC" for these signals.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn028						GR2	GR1	
Gn029		*SSTP	SOR	SAR				
Gn030	SOV7	SOV 6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0
Fn001				ENB				
Fn007						SF		
Fn022	S07	S06	S05	S04	S03	S02	S01	S00

Fn023	S15	S14	S13	S12	S11	S10	S09	S08
Fn024	S23	S22	S21	S20	S19	S18	S17	S16
Fn025	S31	S30	S29	S28	S27	S26	S25	S24
Fn034						GR3O	GR2O	GR1O
Fn036	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
Fn037					R12O	R11O	R10O	R09O

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3703					MPP			2P2
					MPP			

[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 2P2 When a 2-path system is used, inter-path spindle control allows:
0: Configuration where the spindle that belongs to one path only is shared between path 1 and path 2.
1: Configuration where the spindles that belong to path 1 and 2 are shared between the two paths.

#3 MPP In multi-spindle control, a spindle selection using a programmed command instead of using the signals (SWS1 to SWS2<G027.0, 1>) is:
0: Not made.
1: Made.

NOTE
When this parameter is set to 1, set parameter No. 3781 at the same time.

	#7	#6	#5	#4	#3	#2	#1	#0
3705		SFA		EVS	SGT	SGB		ESF
		SFA	NSF		SGT	SGB	GST	ESF

[Input type] Parameter input
[Data type] Bit path

#0 ESF When constant surface speed control is selected (bit 0 (SSC) of parameter No. 8133 is 1) or bit 4 (GTT) of parameter No. 3706 is 1:
0: S codes and SF are output for all S commands.
1: For the T series:
S codes and SF are not output for an S command in the constant surface speed control (G96) mode and a command for maximum spindle speed clamping (G92S-- - ; (G50 for G code system A)).

For the M series:
S codes and SF are not output for an S command in the constant surface speed control (G96) mode.

NOTE
The operation of this parameter varies between the T series and M series.
For the T series: This parameter is valid when bit 4 (EVS) of parameter No. 3705 is set to 1.
For the M series: For an S command for maximum spindle speed clamping (G92S-- - ;), SF is not output, regardless of the setting of this parameter.

#1 GST The SOR signal is used for:
0: Spindle orientation
1: Gear shift

#2 SGB Gear switching method
0: Method A (Parameters 3741 to 3743 for the maximum spindle speed at each gear are used for gear selection.)
1: Method B (Parameters 3751 and 3752 for the spindle speed at the gear switching point are used for gear selection.)

#3 SGT Gear switching method during tapping cycle (G84 and G74)
0: Method A (Same as the normal gear switching method)
1: Method B (Gears are switched during tapping cycle according to the spindle speed set in parameters 3761 and 3762).

#4 EVS S codes and SF are:
0: Not output for an S command.
1: Output for an S command.
The output of S codes and SF for an S command in constant surface speed control mode (G96), or for an S command used to specify maximum spindle speed clamping (G50S---;) depends on the setting of bit 0 (ESF) of parameter No.3705.

#5 NSF For the M series, if T type gear is selected (bit 4 (GTT) of parameter No. 3706 is 1 or constant surface speed control is enabled (bit 0 (SSC) of parameter No. 8133 is 1)), when an S code is specified:
0: SF is output.
1: SF is not output.

NOTE
This parameter does not affect S code output. For an S command for maximum spindle speed clamping (G92S-- - ;), SF is not output, regardless of the setting of this parameter.

#6 SFA The SF signal is output:
0: When gears are switched.
1: Irrespective of whether gears are switched.

	#7	#6	#5	#4	#3	#2	#1	#0
3706	TCW	CWM	ORM		PCS	MPA		
	TCW	CWM	ORM	GTT		MPA		

[Input type] Parameter input
[Data type] Bit path

#2 MPA If a spindle is to be selected using a P command (with bit 3 (MPP) of parameter No. 3703 set to 1) in multi-spindle control, and a P command is not specified together with an S command:

0: The alarm (PS5305) is issued.
1: The last P specified by S_ P_; (by S_ P_; specified for the path in case of a 2-path system) is used. If P is not specified even once after power-up, the value of parameter No. 3775 is used.

NOTE
This parameter is valid only when bit 3 (MPP) of parameter No. 3703 is set to 1.

#3 PCS When multi-spindle control is enabled in each path in a 2-path system, the position coder select signal (PC2SLC<Gn028.7>) to be used is:

0: The signal of a path selected by the inter-path spindle feedback signal.
1: The signal of the local path.

#4 GTT Selection of a spindle gear selection method

0: Type M.
1: Type T.

NOTE
1 M type
The gear selection signal is not input. The CNC selects a gear based on the speed range of each gear set by a parameter beforehand according to S codes, and the selected gear is posted by outputting the gear selection signal. Moreover, the spindle speed matching the gear selected by the output gear selection signal is output.
T type
The gear selection signal is input. The spindle speed matching the gear selected by this signal is output.
2 When constant surface speed control is selected (bit 0 (SSC) of parameter No. 8133 is 1), the T type is assumed regardless of the setting of this parameter.
3 When type T spindle gear switching is selected, the following parameters have no effect:
No.3705#2(SGB), No.3751, No.3752,
No.3705#1(GST), No.3705#3(SGT), No.3761, No.3762,
No.3705#6(SFA), No.3735, No.3736
On the other hand, parameter No. 3744 becomes usable.

#5 ORM Voltage polarity during spindle orientation

0: Positive
1: Negative

#6 CWM

#7 TCW Voltage polarity when the spindle speed voltage is output

TCW	CWM	Voltage polarity
0	0	Both M03 and M04 positive
0	1	Both M03 and M04 negative
1	0	M03 positive, M04 negative
1	1	M03 negative, M04 positive

	#7	#6	#5	#4	#3	#2	#1	#0
3708		TSO					SAT	SAR
		TSO						SAR

[Input type] Parameter input
[Data type] Bit path

#0 SAR The spindle speed arrival signal (SAR) is:
0: Not checked
1: Checked

#1 SAT Check of the spindle speed arrival signal at the start of executing the thread cutting block
0: The signal is checked only when bit 0 (SAR) of parameter No. 3708 is set.
1: The signal is always checked irrespective of whether SAR is set.

NOTE
When thread cutting blocks are consecutive, the spindle speed arrival signal is not checked for the second and subsequent thread cutting blocks.

#6 TSO During a threading or tapping cycle, the spindle override is:
0: Disabled (tied to 100%).
1: Enabled.

NOTE
During rigid tapping, the override is tied to 100%, irrespective of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3709								SAM

[Input type] Parameter input
[Data type] Bit path

#0 SAM The sampling frequency to obtain the average spindle speed
0: 4 (Normally, set to 0.)
1: 1

	#7	#6	#5	#4	#3	#2	#1	#0
3712		GMB						

[Input type] Parameter input
[Data type] Bit

#6 GMB With type-M gear switching method B, the speed of each gear is clamped to:

0 : The maximum rotation speed (No. 3741 to No. 3743) of each gear or the maximum clamping speed (No. 3736) of the spindle motor.

1 : The spindle motor speed which is different every gear.

Gear1 : The spindle motor speed (No. 3751) at the gear switching point between gear 1 and gear 2

Gear2 : The spindle motor speed (No. 3752) at the gear switching point between gear 2 and gear 3

Gear3 : The maximum clamping speed (No. 3736) of the spindle motor

	#7	#6	#5	#4	#3	#2	#1	#0
3715								NSAx

[Input type] Parameter input
[Data type] Bit axis

#0 NSAx When a move command is executed for an axis, the spindle speed arrival signal SAR is:

0: Checked.

1: Not checked.

Set an axis for which the spindle speed arrival signal SAR need not be checked when a move command is executed for the axis. When a move command is specified only for an axis with this parameter set to 1, the spindle speed arrival signal SAR is not checked.

	#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 To use a serial spindle, set bit 5 (SSN) of parameter No. 8133 to 0.
2 A maximum of one analog spindle can be controlled.
3 When using an analog spindle, set it at the end of the spindle configuration.

3717	Spindle amplifier 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 spindles

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.
3: Spindle motor connected to amplifier number 3 is used.

NOTE
When using an analog spindle, set it at the end of the spindle configuration.
(Example)
When there are three spindles in an entire system (two serial spindles and one analog spindle), set the spindle amplifier number (this parameter) of the analog spindle to 3.

3718	Subscript for display of a serial spindle or analog spindle

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 122
Set a subscript to be added to spindle speed display on a screen such as the position display screen.

3720	Number of position coder pulses

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] Detection unit
[Valid data range] 1 to 32767
Set the number of position coder pulses.

3721	Number of gear teeth on the position coder side

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 9999
Set the number of gear teeth on the position coder side in speed control (such as feed per revolution and threading).

3722	Number of gear teeth on the spindle side

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 9999
Set the number of gear teeth on the spindle side in speed control (such as feed per revolution and threading).

3730	Data used for adjusting the gain of the analog output of spindle speed

[Input type] Parameter input

[Data type] Word spindle
[Unit of data] 0.1%
[Valid data range] 700 to 1250
Set data used for adjusting the gain of the analog output of spindle speed.
[Adjustment method] <1> Assign standard value 1000 to the parameter.
<2> Specify the spindle speed so that the analog output of the spindle speed is the maximum voltage (10 V).
<3> Measure the output voltage.
<4> Assign the value obtained by the following equation to parameter No.3730.
Setting value = (10 (V) / Measured data (V)) × 1000
<5> After setting the parameter, specify the spindle speed so that the analog output of the spindle speed is the maximum voltage. Confirm that the output voltage is 10V.

NOTE
This parameter needs not to be set for serial spindles.

3731 **Compensation value for the offset voltage of spindle speed analog output**

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Velo
[Valid data range] -1024 to 1024
Set a compensation value for the offset voltage of spindle speed analog output.
Setting = -8191 × offset voltage (V)/12.5
[Adjustment method] <1> Assign standard value 0 to the parameter.
<2> Specify the spindle speed so that the analog output of the spindle speed is 0.
<3> Measure the output voltage.
<4> Assign the value obtained by the following equation to parameter No.3731.
Setting value = (-8191 × Offset voltage (V)) / 12.5
<5> After setting the parameter, specify the spindle speed so that the analog output of the spindle speed is 0. Confirm that the output voltage is 0V.

NOTE
This parameter needs not to be set for serial spindles.

3732 **The spindle speed during spindle orientation or the spindle motor speed during spindle gear shift**

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 20000
Set the spindle speed during spindle orientation or the spindle motor speed during gear shift.
When bit 1 (GST) of parameter No. 3705 is set to 0, set the spindle speed during spindle orientation in min^{-1}.
When bit 1 (GST) of parameter No. 3705 is set to 1, set the spindle motor speed during spindle gear shift calculated from the following formula.
For a serial spindle
Setting value =
(Spindle motor speed during spindle gear shift / Maximum spindle motor speed) × 16383

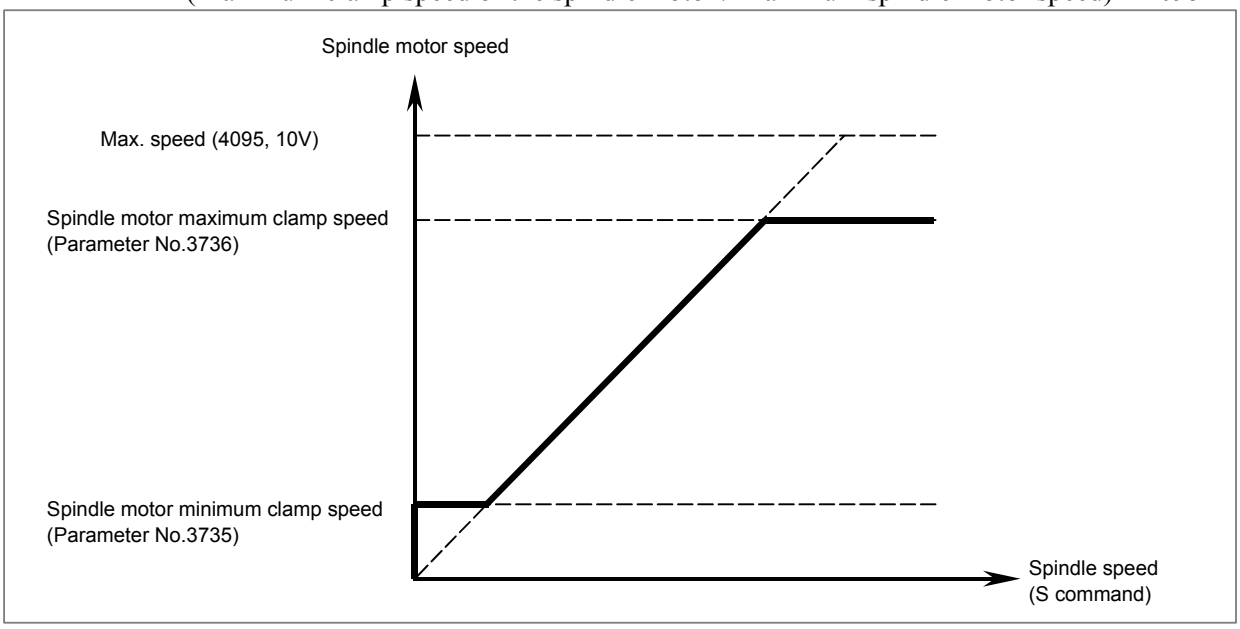
For an analog spindle
Setting value =
(Spindle motor speed during spindle gear shift / Maximum spindle motor speed) × 4095

3735	
	Minimum clamp speed of the spindle motor

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 4095
Set the minimum clamp speed of the spindle motor.
Setting value =
(Minimum clamp speed of the spindle motor / Maximum spindle motor speed) × 4095

3736	
	Maximum clamp speed of the spindle motor

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 4095
Set the maximum clamp speed of the spindle motor.
Setting value =
(Maximum clamp speed of the spindle motor / Maximum spindle motor speed) × 4095

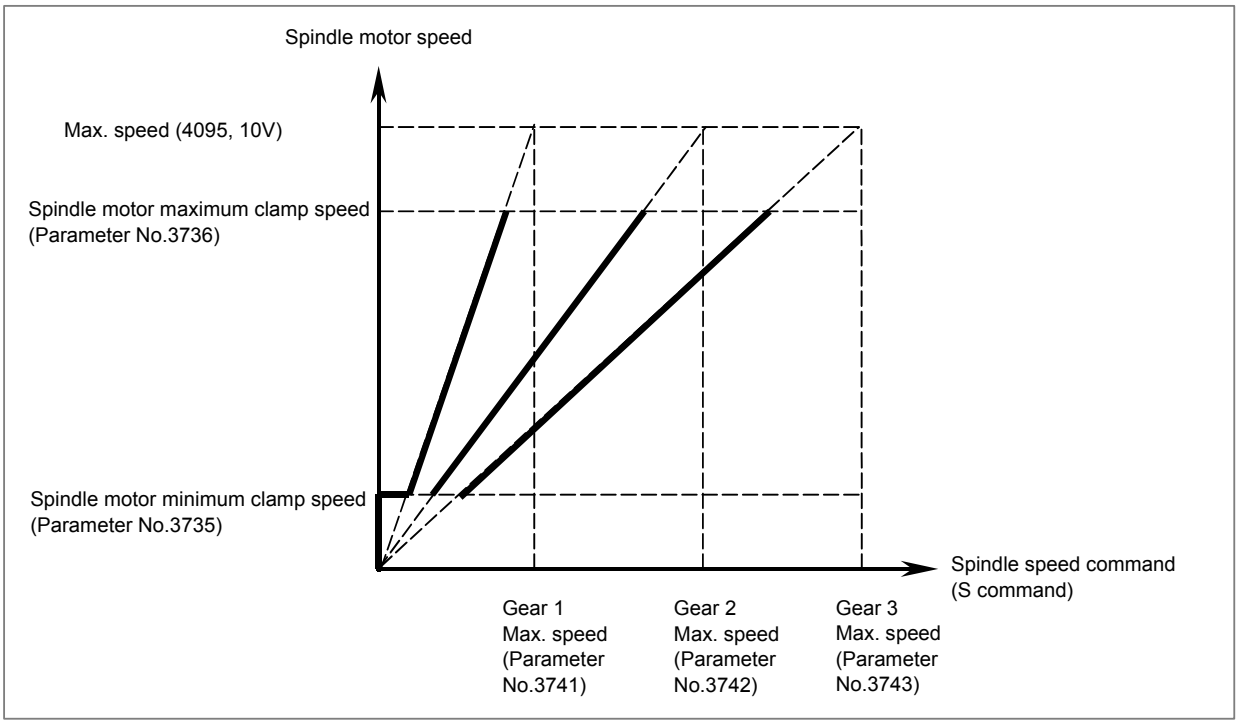


3740	**Time elapsed prior to checking the spindle speed arrival signal**

[Input type] Parameter input
[Data type] Word path
[Unit of data] msec
[Valid data range] 0 to 32767
Set the time elapsed from the execution of the S function up to the checking of the spindle speed arrival signal.

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.

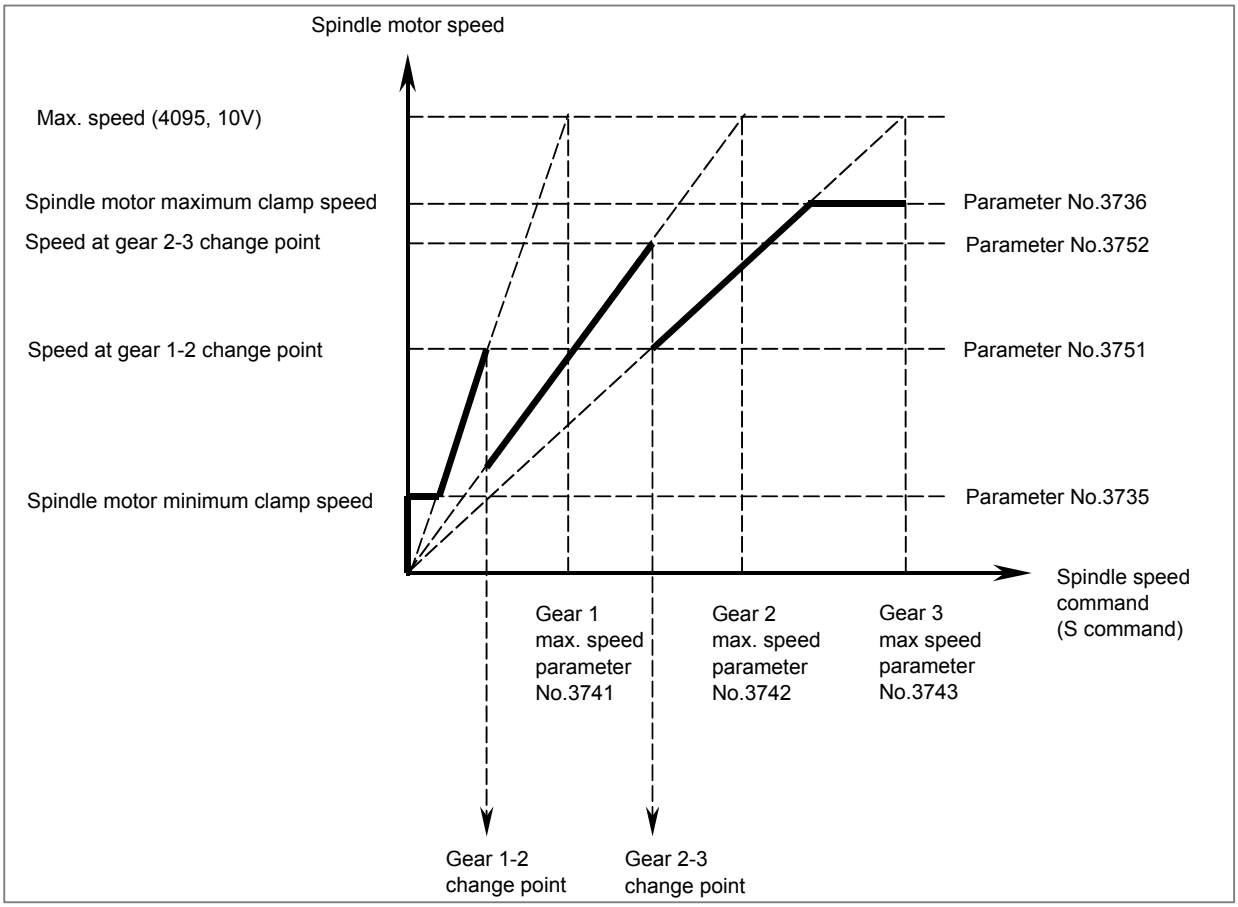


3751	
	Spindle motor speed when switching from gear 1 to gear 2

3752	
	Spindle motor speed when switching from gear 2 to gear 3

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 4095

For gear switching method B, set the spindle motor speed when the gears are switched.
Setting value =
(Spindle motor speed when the gears are switched / Maximum spindle motor speed) × 4095

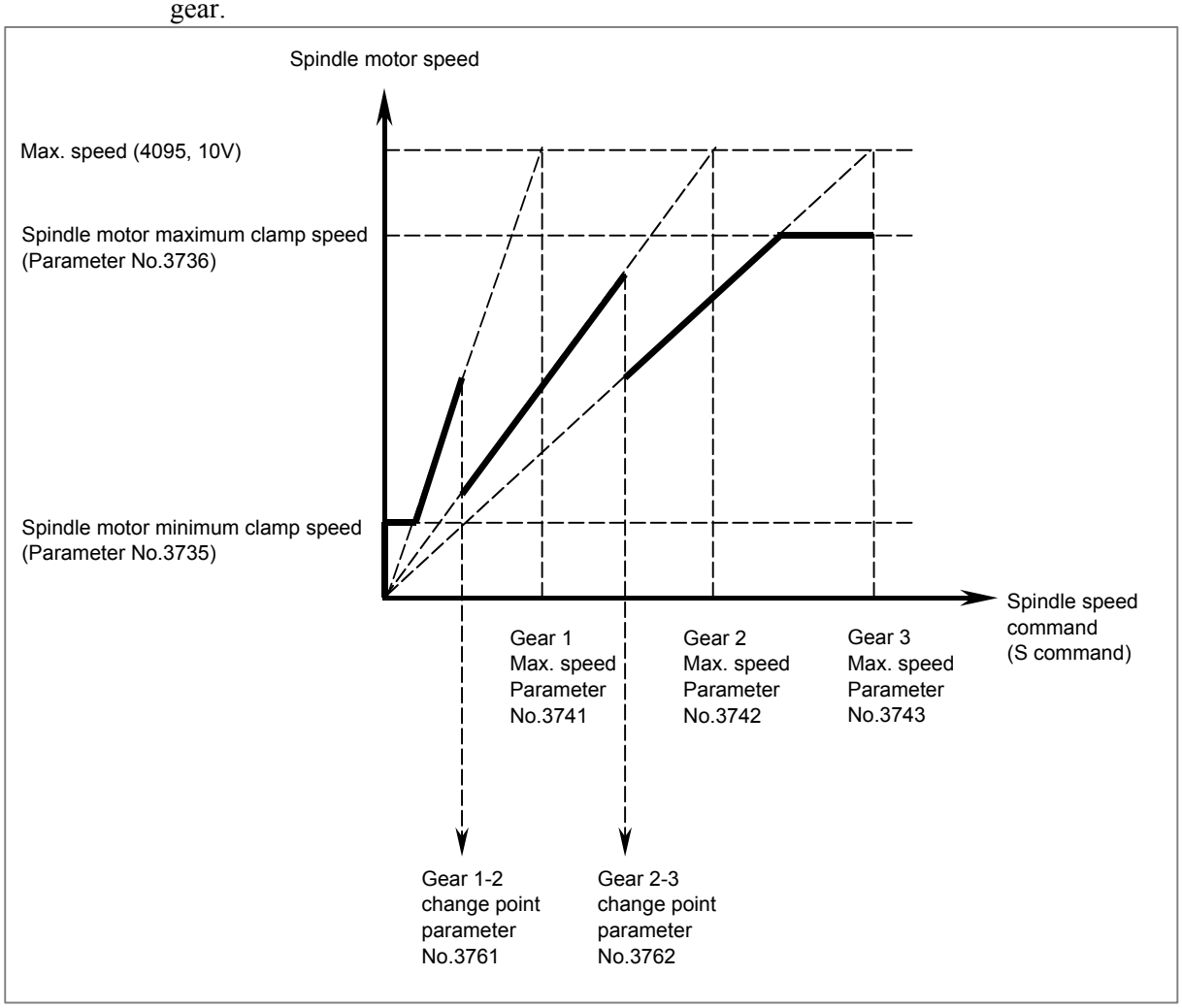


3761	
	Spindle speed when switching from gear 1 to gear 2 during tapping cycle

3762	
	Spindle speed when switching from gear 2 to gear 3 during tapping cycle

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

When method B is selected as the gear change method in the tapping cycle (when bit 3 (SGT) of parameter No. 3705 is set to 1), set the spindle speed at a change point of each gear.



3772	**Maximum speed for each spindles**

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

This parameter sets the maximum spindle speed.
When a command specifying a speed exceeding the maximum speed of the spindle is specified, or the speed of the spindle exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

⚠ CAUTION
1 When 0 is set in this parameter, the speed of the spindle is not clamped.
2 When spindle speed command control is applied using the PMC, this parameter has no effect, and the spindle speed is not clamped.

NOTE
1 For the M series, this parameter is valid when constant surface speed control is selected (bit 0 (SSC) of parameter No. 8133 is 1).
2 When constant surface speed control is provided (bit 0 (SSC) of parameter No. 8133 is set to 1), clamping at the maximum speed is performed in either of the G96 and G97 modes.

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.

NOTE
1 This parameter is valid if bit 3 (MPP) of parameter No. 3703 is set to 1.
2 If this parameter is set to 0, the corresponding spindle cannot be selected by a P code.
3 Under 2-path control, the P code specified here is valid for each path.
For instance, if the P code to select the first spindle of path 2 is set to 21, specifying S1000 P21; in path 1 causes the first spindle of path 2 to be rotated at S1000.
4 Identical P code values cannot be used for different spindles. (Identical P code values cannot be used even if the paths are different.)
5 When this parameter is used (when bit 3 (MPP) of parameter No. 3703 is set to 1), the spindle command selection signal is invalid.
6 To use this parameter, enable multi-spindle control (bit 3 (MSP) of parameter No. 8133 is 1).

Caution

⚠ CAUTION
This section mentioned a spindle speed control that should be prepared on the CNC side. But it is also necessary to design the processing of the signals to the spindle control unit.
Refer to the manual of the spindle control unit used and take necessary actions on the spindle control unit.

10.7 SPINDLE OUTPUT CONTROL BY THE PMC

Overview

The PMC can control the speed and polarity of each spindle motor, connected.
The first to second spindles each have their own individual interfaces. By using a PMC ladder program, the user can control the spindles as desired. This section describes how to use the PMC to control spindle rotation and provides example applications.

Explanation

- Switching control

This function can be used to specify the following:

- Spindle motor speed (number of rotations)
- Output polarity for each spindle motor (direction of rotation)

Usually, the CNC is used to control the speed and polarity of the first spindle motor. When the multi-spindle control function is provided (bit 3 (MSP) of parameter No. 8133 is set to 1), the second spindle is also placed under the CNC's spindle control.
This function allows the user to select whether the CNC or PMC is used to control the speed and output polarity of the spindle motors.

- Specifying the spindle motor speed

The PMC can be used to specify the spindle motor speed upon executing the following:

- Switching the controller from the CNC to the PMC, by issuing SINDx signal
- Setting the spindle motor speed data, calculated by the PMC, in spindle control signals R01Ix to R12Ix

When controlled by the PMC, the spindle motor speed is not affected by any signal (for example, the spindle speed override signal) or parameter settings (for example, the maximum speed clamp parameter) related to the spindle speed command of the CNC spindle control function.

→ Exceptionally, when multi-spindle control is provided (bit 3 (MSP) of parameter No. 8133 is set to 1), a spindle stop signal *SSTPx<Gn027.3 or Gn027.3.4> stops the corresponding spindle even if it is being controlled by the PMC.

The spindle motor speed data is obtained from the following expression. Its value can range from 0 to 4095:

Spindle motor speed data =

(Spindle motor speed/Maximum spindle motor speed (parameter No. 4020))×4095

Remark)

Usually, the spindle speed must be controlled. If a gear train is used to connect the spindle to the spindle motor, first obtain the maximum spindle speed at the maximum spindle motor speed.

Spindle motor speed data =

(Spindle speed/Maximum spindle speed)×4095

By using this expression, the spindle motor speed data can easily be obtained.

- Specifying the output polarity for the spindle motor

The PMC can specify the spindle motor output polarity when the following are executed:

- Switching the controller from the CNC to the PMC, by issuing an SSINx signal
- Specifying the output polarity to the SGNx signal

- S-code and SF signals

To control the spindle, the PMC may be required to read the S value specified by the CNC.
If the spindle serial output/spindle analog control function is added (if the PMC can control the spindle), the S-code signals <Fn022 to Fn025> and SF signal <Fn007.2> can be output only when several conditions, determined by the CNC spindle control, are satisfied. In some cases, the signals cannot be used under standard conditions.
Specify the related bits of parameter No. 3705 according to the desired application, then use the S-code and SF signals.

- Twelve code signals corresponding to the S value (output)

Twelve code signals corresponding to S value R01O to R12O <Fn036.0 to Fn037.3> are output to the first spindle motor. The output data is calculated from the results of the CNC spindle control. (See Section SPINDLE SPEED CONTROL.)
Even while a spindle is subject to PMC control, an S command that is issued to the CNC is converted to spindle output data and output.
The SIND signal determines whether the speed output command, issued to the spindle motor, is obtained from the twelve code signals corresponding to the S value, or from the R01I to R12I signals calculated and specified by the PMC.
The use of this signal may simplify PMC ladder processing used to enable PMC spindle control.
Using twelve code signals corresponding to S values R01O to R12O <Fn036.0 to Fn037.3>, the input command status of spindle output control by the PMC is not output.

Example

- Sample application 1)

Controlling the first and second spindles of a T series

→ Share the gear stages between the first and second spindles.
(If the first spindle uses two gears, for example, specify parameters Nos. 3743 and 3744, thus enabling the use of gears 3 and 4 for the second spindle.)
Perform the necessary setting to enable control of the first and second spindles by the PMC.
To specify a rotation command for the first spindle, enter the gears for the first spindle in GR1 and GR2 and obtain the data of the twelve code signals corresponding to the S value. Specify the data as the speed output command for the first spindle in the PMC control interface for the first spindle.
To specify a rotation command for the second spindle, enter the gears to be used for the second spindle in GR1 and GR2 and obtain the data of the twelve code signals corresponding to the S value. Specify the data as the speed output command for the second spindle in the PMC control interface for the second spindle.

- Sample application 2)

With the stop position of the serial spindle specified externally, specifying the S value as the angle of the stop position for spindle orientation after the spindle positioning mode has been selected

→ Use the gears that are not being used for the first spindle.
(In this application, gear 4 is used to calculate the spindle position. Set parameter No. 3744 to 360.)
Specify the M code used to set the spindle to positioning mode and stop the spindle. Enter gear 4 in GR1 and GR2.
Then, specify a spindle positioning angle with the S command.
(To specify the position of 145 degrees, for example, specify S145;.)
Expression 145/360 4095 is calculated and the result is output to the twelve code signals corresponding to the S value (output signal). Enter the data in external stop position commands SHA00 to SHA11 <Gn078.0 to Gn079.3> and perform the orientation.

Signal

PMC spindle control signals

SIND,SSIN,SGN,<Gn033.7,6,5>R01I to R12I<Gn032.0 to Gn033.3> : for the first spindle

SIND2,SSIN2,SGN2,<Gn035.7,6,5>R01I2 to R12I2<Gn034.0 to Gn035.3> : for the second spindle

[Classification] Input signal

[Function] The above signals enable the control of a spindle motor by issuing commands from the PMC.
Both the speed and polarity of the spindle motor (direction of rotation) can be controlled.
The speed command and polarity are usually specified by the CNC. The use of these signals allows the user to select whether the speed and polarity are controlled by the CNC or PMC.
Even when multi-spindle control is not provided (bit 3 (MSP) of parameter No. 8133 is set to 0), the second spindle can be controlled with this signal.
When multi-spindle control is provided (bit 3 (MSP) of parameter No. 8133 is set to 1) and Type A is set (bit 2(MSI) of parameter No. 3709 is set to 0), the signal for the second spindle cannot be used.

- Details of the signals

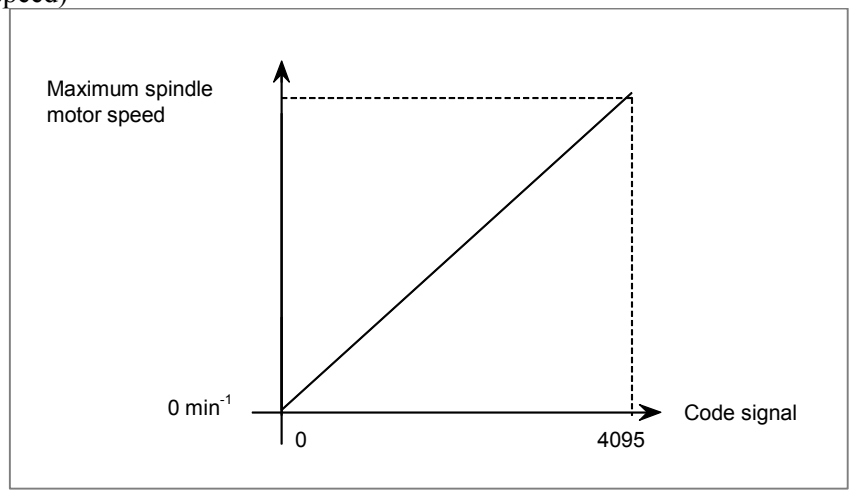
- Signal used to select the spindle motor speed command SINDx

→ The above signal is used to select whether the spindle motor speed is controlled by the CNC or PMC.
"1" : The spindle motor is controlled according to speed commands (R01Ix to R12Ix) issued by the PMC.
"0" : The spindle motor is controlled according to speed commands issued by the CNC. The spindle speed specified with the S command is output.

- Signals used to input the spindle motor speed command issued by the PMC R01Ix to R12Ix

→ If the PMC is being used to control the spindle motor speed command, specify, in binary format, the value obtained using the following expression.
Value to be specified =
(Spindle motor speed/
Maximum spindle motor speed (parameter No. 4020))×4095

(Spindle motor speed)



- Signal used to select the polarity of the spindle motor speed command, SSINx

→ The above signal selects whether the output polarity of the spindle motor speed command is controlled by the CNC or PMC.
"1" : The spindle motor is controlled according to the polarity command (SGNx) issued by the PMC.
"0" : The CNC controls the polarity. The polarity is determined by the parameter TCW and CWM bits (No. 3706#7 and #6) and the M03 or M04 command.

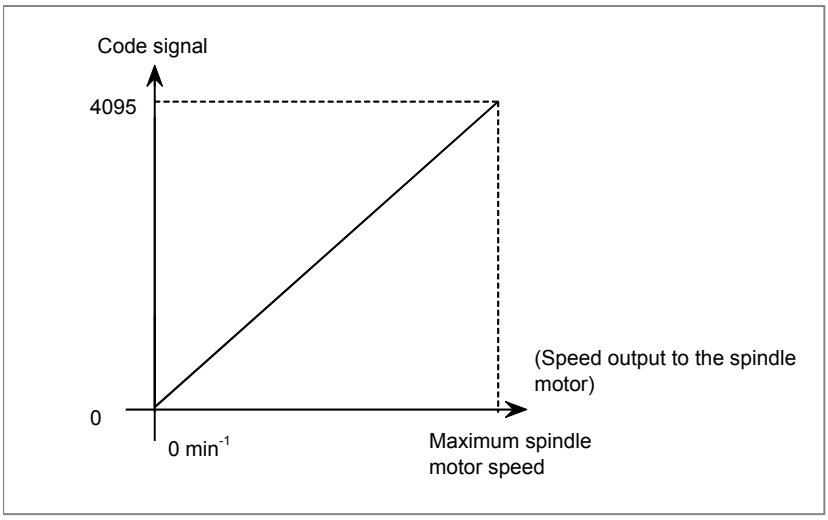
- Signal used to specify the polarity of the spindle motor selected by the PMC, SGNx

→ If the PMC is used to control the output polarity of the spindle motor speed command, specify the polarity with this signal.
"1" : The output polarity of the spindle is negative.
"0" : The output polarity of the spindle is positive.

Twelve code signals corresponding to the S value R01O to R12O<Fn036.0 to Fn037.3>

[Classification] Output signal

[Function] The S value, specified in the CNC part program, is converted to the speed output of the spindle motor that is required to control the connected spindle. The converted value is sent to the PMC with twelve code signals, in proportional to the spindle motor speed output.
The speed data, the final result of the CNC spindle control, is output to the spindle motor after the spindle gear ratio, spindle speed override, speed clamp, conversion of the surface speed into the spindle speed by the constant surface speed control command, and other data have been considered.



Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn032	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
Gn033	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
Gn034	R08I2	R07I2	R06I2	R05I2	R04I2	R03I2	R02I2	R01I2
Gn035	SIND2	SSIN2	SGN2		R12I2	R11I2	R10I2	R09I2

	#7	#6	#5	#4	#3	#2	#1	#0
Fn036	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
Fn037					R12O	R11O	R10O	R09O

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3705		SFA		EVS				ESF
		SFA	NSF					ESF

[Input type] Parameter input
[Data type] Bit path

#0 ESF When constant surface speed control is selected (bit 0 (SSC) of parameter No. 8133 is 1) or bit 4 (GTT) of parameter No. 3706 is 1:
0: S codes and SF are output for all S commands.
1: For the T series:
S codes and SF are not output for an S command in the constant surface speed control (G96) mode and a command for maximum spindle speed clamping (G92S---- ; (G50 for G code system A)).
For the M series:
S codes and SF are not output for an S command in the constant surface speed control (G96) mode.

NOTE
The operation of this parameter varies between the T series and M series.
For the T series: This parameter is valid when bit 4 (EVS) of parameter No. 3705 is set to 1.
For the M series: For an S command for maximum spindle speed clamping (G92S-- - ;), SF is not output, regardless of the setting of this parameter.

#4 EVS S codes and SF are:
0: Not output for an S command.
1: Output for an S command.
The output of S codes and SF for an S command in constant surface speed control mode (G96), or for an S command used to specify maximum spindle speed clamping (G50S---;) depends on the setting of bit 0 (ESF) of parameter No.3705.

#5 NSF For the M series, if T type gear is selected (bit 4 (GTT) of parameter No. 3706 is 1 or constant surface speed control is enabled (bit 0 (SSC) of parameter No. 8133 is 1)), when an S code is specified:
0: SF is output.
1: SF is not output.

NOTE
This parameter does not affect S code output. For an S command for maximum spindle speed clamping (G92S-- - ;), SF is not output, regardless of the setting of this parameter.

#6 SFA The SF signal is output:
0: When gears are switched.
1: Irrespective of whether gears are switched.

	#7	#6	#5	#4	#3	#2	#1	#0
3709						MSI		

[Input type] Parameter input
[Data type] Bit path

#2 MSI In multi-spindle control, the SIND signal is valid
0: Only when the first spindle is valid (SIND signal for the 2nd, 3rd spindle becomes ineffective) (TYPE-A)
1: For each spindle irrespective of whether the spindle is selected (Each spindle has its own SIND signal). (TYPE-B)

Note

NOTE
1 If the spindle fails to move after the PMC issues the spindle motor speed command, check the following:
Is TYPE-A set (is bit 2 (MSI) of parameter No. 3709 set to 0) when multi-spindle control is provided (bit 0 (SSC) of parameter No. 8133 is set to 1)?
→ The second spindle cannot be controlled. The first spindle can be controlled only when the spindle selection signal SWS1 is set to "1".
Is the spindle stop signal for each axis set to 0 when multi-spindle control is provided (bit 0 (SSC) of parameter No. 8133 is set to 1)?
→ Spindle stop signal for each axis *SSTPx <G027.3, 4> stops the spindle.
M03/M04 is not specified when the CNC is being used to control the output polarity.
→ If the TCW bit, bit 7 of parameter No. 3706, is set to 1, the M03/M04 command issued to the CNC changes the output polarity for the spindle motor. If no M03/M04 command is specified after the CNC is turned on, the specified speed output is not sent to the spindle motor because the output polarity has not been determined.
2 The SF signal indicates that output of the S code to the PMC has been completed. The signal does not indicate the end of the command for specifying the spindle speed.
3 For an explanation of connecting the second spindle, see Sections "SPINDLE SERIAL OUTPUT" and "MULTI-SPINDLE CONTROL."
4 When multi-spindle control is not provided (bit 0 (SSC) of parameter No. 8133 is set to 0), no command is issued from the CNC to the second spindle, so the output polarity is controlled by the SGNx signal regardless of the SSINx signal status.
The speed output to the spindle motor can be controlled only when the SINDx signal is set to "1".

10.8 CONSTANT SURFACE SPEED CONTROL

Overview

When a surface speed (in m/min or feet/min) is specified with an S code (a numeric value following S), the spindle speed is controlled so that the surface speed is kept constant with respect to the change in tool position.

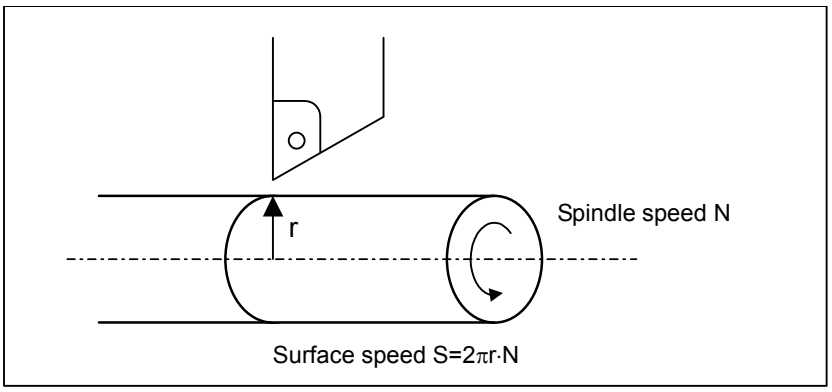


Fig. 10.8 (a) Relationship between surface speed and spindle speed

NOTE
When using constant surface speed control, set bit 0 (SSC) of parameter No. 8133 to 1.

Explanation

To enable or disable constant surface speeed control, use a G code.
Specify constant surface speed control as follows:

G96 S_ (Surface speed m/min (in metirci input) or feet/min (in inch input))

To cancel constant surface speed control, specify the following:

G97 S_ (Spindle speed min^{-1})

The axis to be placed under constant surface speed control is specified as follows:

G96 Pα (P0: Axis set in parameter No. 3770, P1 to Pn: First axis to n-th axis (axes are set with numbers within a path))

To perform constant surface speed control, the maximum spindle speed for each gear select signal issued from the PMC side must be set by parameter No. 3741-3744.
The gear select signal is a 2 bit code signal (GR1, GR2). The relationship between the signal and gear number is :

GR1	GR2	Gear number
0	0	1
1	0	2
0	1	3
1	1	4*

(* :Gear 4 is available only in the lathe specifications.)

- Constant surface speed control in rapid traverse (G00)

When a rapid-traverse block specified by G00 is executed, the constant surface speed control function calculates the surface speed based on the position of the end point of the block from the beginning, instead of calculating the surface speed for the tool position that changes from time to time.
If bit 1 (RSC) of parameter No. 8709 is set to 1, however, the surface speed for such rapid-traverse blocks is calculated according to the tool position that changes from time to time in the same manner as for cutting feed blocks.

- Constant surface speed control when a machine lock is applied

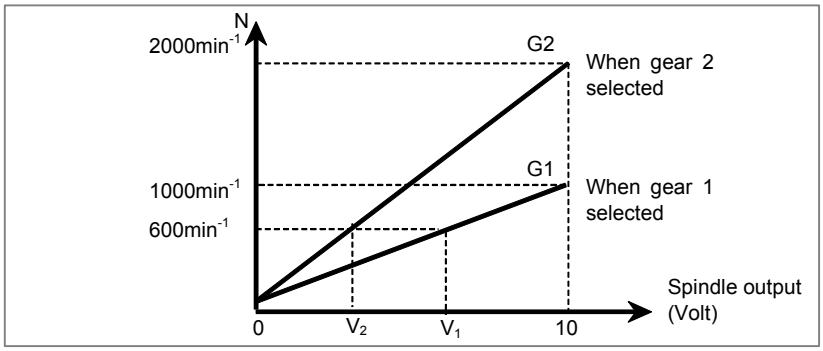
When a machine lock is applied, the surface speed is calculated according to the change in programmed coordinates although the machine does not move.

- Selecting the axis to be placed under constant surface speed control

The axis to be placed under constant surface speed control can be specified in parameter No. 3770 or by programming.
When the target axis for constant surface speed control is specified by programming, address P of a G96 block is used. If P is omitted, or if P0 is specified, the axis set in parameter No. 3770 is assumed.
If parameter No. 3770 is set to 0, however, any programmed axis is ignored, and constant surface speed control is performed with respect to the X-axis.

- Example of Spindle Analog Output

Assume that gear switching is two stage switching. If the spindle speed with the output 10 V is 1000 min^{-1} for the low speed gear (G1) and 2000 min^{-1} for the high speed gear (G2), set these speeds to the parameter No. 3741, 3742, respectively. In this case, the spindle output has the linear relationship shown below:



Here, S = 60 m/min is given as the surface speed; if the position of the present X-axis cutter is 16 mm from the center, the spindle speed N becomes 600 min^{-1} ($S = 2\pi r \cdot N$). Therefore, V_1 (for G1) or V_1 (for G2) is calculated inside the CNC and output to the machine side.

V1 ; 6 (V)
V2 ; 3 (V)

The value of output voltage V is calculated automatically from the following equations:

(i) G96

$V=10S/2\pi rR$

R : Spindle speed (min^{-1}) at 10V output voltage (that is , spindle speed set by parameter No. 3741 to No. 3744)
S : Surface speed (m/min) specified by S command
r : Radius value in the X-axis direction (m)

(ii) G97

$V=10N/R$

R : Spindle speed at 10V output voltage (min^{-1})
N : Spindle speed given by S command (min^{-1})

Spindle Serial Output

The output to the spindle in spindle serial output is a digital data.
Therefore assume the following relation for calculation:
Spindle analog output (voltage) 10V = Spindle serial output (digital data) 4095.
The above calculation becomes as follows:
The value of Spindle output D:

(i) G96

$D=4095S/2\pi rR$

R : Spindle speed (min^{-1}) at maximum spindle motor speed (that is , spindle speed set by parameter No. 3741 to No. 3744)
S : Surface speed (m/min) specified by S
r : Radius value in the X-axis direction (m)

(ii) G97
D=4095N/R
R : Spindle speed at maximum spindle motor speed (min^{-1})
N : Spindle speed given by S command (min^{-1})

Constant surface speed control with no position coder

With a machine having (or using) no position coder, constant surface speed control cannot usually be performed during feed per revolution, but is enabled by setting bit 1 (FPR) of parameter No. 3729 or bit 2 (PCL) of parameter No. 1405 to 1.

For example, the following program is executed with G code system B and diameters specified.

```
N1  G00  X50.  Z10. ;
N2  G96  G95  S12 ;        ← Constant surface speed control and feed per revolution begin.
N3  G01  X20.  Z30. F10.;
N4  Z50. ;
N5  G97  S200;             ← Constant surface speed control ends.
N6  G00  Z200.;
N7  M30 ;
```

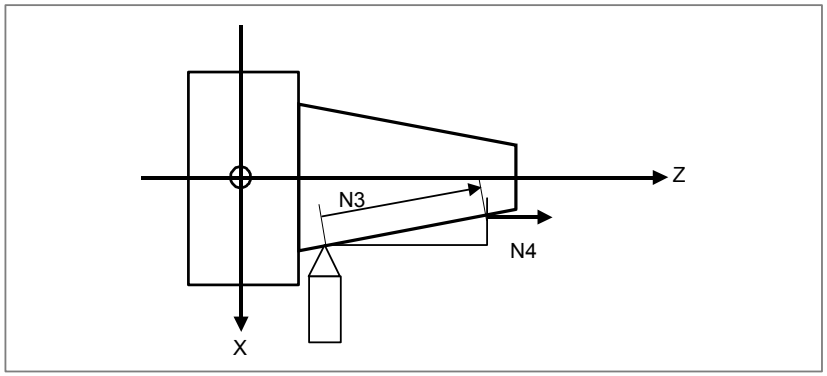


Fig. 10.8 (b) Feed per revolution without position coder

In this program, block N2 issues a constant surface speed control command (G96), a surface speed command (S12 m/min), and a feed-per-revolution command (G95). Block N3 causes the CNC to change the spindle speed specification from 76.4 min^{-1} to 191 min^{-1} so as to maintain a constant surface speed during movement to X = 20.
Meanwhile, the feed-per-revolution speed is changed according to the changed spindle speed specification, and used for movement along the feed axis. However, the specified spindle speed is clamped to the upper limit to the spindle speed specification (for the first spindle, using parameter No. 3772). In the above program, the feed-per-revolution speed command F10 in block N3 corresponds to an actual speed of 764 (mm/min) to 1910 (mm/min).

Signal

Gear selection signal (Input) GR2,GR1<Gn028.2,1>

Refer to "SPINDLE SPEED CONTROL."

Constant surface speed signal CSS<Fn002.2>

[Classification] Output signal
[Function] This signal indicates that constant surface speed control is in progress.
[Output cond.] "1" indicates that constant surface speed control mode (G96) is in progress, while "0" indicates it is not.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn002						CSS		

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1402								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

NOTE
1 When using the position coder, set this parameter to 0.
2 While this parameter is set to 1, threading cannot be performed even if a position coder is provided.

	#7	#6	#5	#4	#3	#2	#1	#0
1405						PCL		

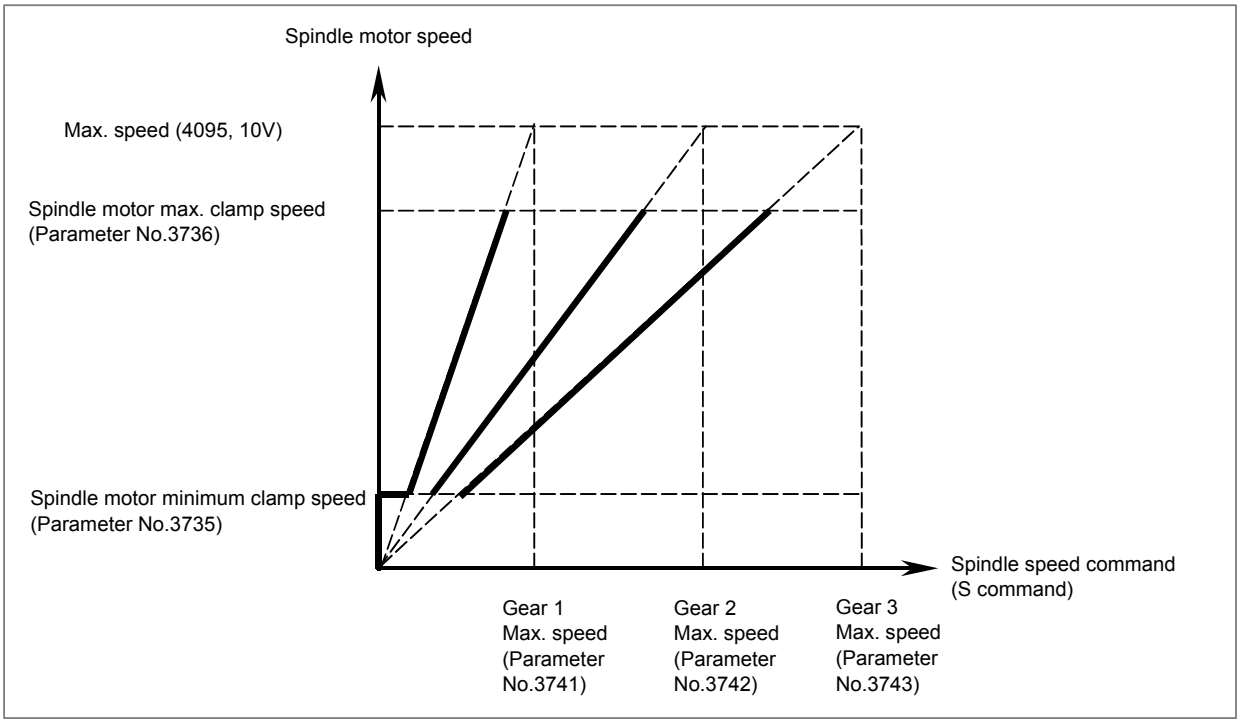
[Input type] Parameter input
[Data type] Bit path

#2 PCL The function for constant surface speed control without the position coder is:
0: Not used.
1: Used.
Used.

NOTE
1 Enable constant surface speed control (set bit 0 (SSC) of parameter No. 8133 to 1).
2 When setting this parameter to 1, set bit 0 (NPC) of parameter No. 1402 to 0.

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 (Note)

[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.



NOTE
When a type-T gear shift scheme is selected for the M series (when constant surface speed control is provided (bit 0 (SSC) of parameter No. 8133 is set to 1) or bit 4 (GTT) of parameter No. 3706 is set to 1), parameter No. 3744 can be used also in the M series.
Note, however, that, even in this case, only up to three main gear stages are usable for rigid tapping.

	#7	#6	#5	#4	#3	#2	#1	#0
3705				EVS				ESF
			NSF					ESF

[Input type] Parameter input
[Data type] Bit path

#0 ESF When constant surface speed control is selected (bit 0 (SSC) of parameter No. 8133 is 1) or bit 4 (GTT) of parameter No. 3706 is 1:
0: S codes and SF are output for all S commands.
1: For the T series:
S codes and SF are not output for an S command in the constant surface speed control (G96) mode and a command for maximum spindle speed clamping (G92S---- ; (G50 for G code system A)).
For the M series:
S codes and SF are not output for an S command in the constant surface speed control (G96) mode.

NOTE
The operation of this parameter varies between the T series and M series.
For the T series: This parameter is valid when bit 4 (EVS) of parameter No. 3705 is set to 1.
For the M series: For an S command for maximum spindle speed clamping (G92S-- - ;), SF is not output, regardless of the setting of this parameter.

#4 EVS S codes and SF are:
0: Not output for an S command.
1: Output for an S command.
The output of S codes and SF for an S command in constant surface speed control mode (G96), or for an S command used to specify maximum spindle speed clamping (G50S---;) depends on the setting of bit 0 (ESF) of parameter No.3705.

#5 NSF For the M series, if T type gear is selected (bit 4 (GTT) of parameter No. 3706 is 1 or constant surface speed control is enabled (bit 0 (SSC) of parameter No. 8133 is 1)), when an S code is specified:
0: SF is output.
1: SF is not output.

NOTE
This parameter does not affect S code output. For an S command for maximum spindle speed clamping (G92S-- - ;), SF is not output, regardless of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3706								
				GTT				

[Input type] Parameter input
[Data type] Bit path

#4 GTT Selection of a spindle gear selection method
0: Type M.
1: Type T.

NOTE
1 M type
The gear selection signal is not input. The CNC selects a gear based on the speed range of each gear set by a parameter beforehand according to S codes, and the selected gear is posted by outputting the gear selection signal. Moreover, the spindle speed matching the gear selected by the output gear selection signal is output.
T type
The gear selection signal is input. The spindle speed matching the gear selected by this signal is output.

NOTE
2 When constant surface speed control is selected (bit 0 (SSC) of parameter No. 8133 is 1), the T type is assumed regardless of the setting of this parameter.
3 When type T spindle gear switching is selected, the following parameters have no effect:
No.3705#2(SGB), No.3751, No.3752,
No.3705#1(GST), No.3705#3(SGT), No.3761, No.3762,
No.3705#6(SFA), No.3735, No.3736
On the other hand, parameter No. 3744 becomes usable.

	#7	#6	#5	#4	#3	#2	#1	#0
3708			SOC					

[Input type] Parameter input
[Data type] Bit path

#5 SOC During constant surface speed control (G96 mode), the speed clamp by the maximum spindle speed clamp command (M series: G92 S_; T series: G50 S_;) is carried out:
0: Before spindle speed override.
1: After spindle speed override.
If this parameter is set to 0, the spindle speed may exceed the maximum spindle speed (numeric value following S in G92 S_; (M series) or G50 S_; (T series)).
If this parameter is set to 1, the spindle speed is limited to the maximum spindle speed.
The spindle speed is limited to the upper limit of spindle speed specified in parameter No. 3772, irrespective of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3709							RSC	

[Input type] Parameter input
[Data type] Bit path

#1 RSC In the constant surface speed control mode, the surface speed of a rapid traverse block is calculated:
0: In accordance with the coordinates of the end point.
1: In accordance with the current value, as in cutting feed.

	#7	#6	#5	#4	#3	#2	#1	#0
3729							FPRs	

[Input type] Parameter input
[Data type] Bit spindle

#1 FPRs Feed per revolution (without a position coder) is:
0: Not used for a spindle.
1: Used for a spindle.
In a machine that does not use a position coder, when bit 1 (FPR) of parameter No. 3729 is set to 1 for each axis, feed per revolution can be performed with a spindle command. A feed per revolution is specified with G95 (G99 for T series) in the same way as for normal operation.
When multispindle control is performed, the target spindle for feed per revolution is selected with a position coder select signal (PC2SLC<Gn028.7>).

NOTE
Enable constant surface speed control (bit 0 (SSC) of parameter No. 8133 is 1).

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.

3771	Minimum spindle speed in constant surface speed control mode (G96)

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min^{-1}
[Valid data range] 0 to 32767
Set the minimum spindle speed in the constant surface speed control mode (G96).
The spindle speed in constant surface speed control is clamped to the speed given by parameter 3771.

3772	Maximum spindle speed

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999
This parameter sets the maximum spindle speed.
When a command specifying a speed exceeding the maximum speed of the spindle is specified , or the speed of the spindle exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

⚠ CAUTION
1 When 0 is set in this parameter, the speed of the spindle is not clamped.
2 When spindle speed command control is applied using the PMC, this parameter has no effect, and the spindle speed is not clamped.

NOTE
1 For the M series, this parameter is valid when constant surface speed control is selected (bit 0 (SSC) of parameter No. 8133 is 1).
2 When the constant surface speed control is selected, the spindle speed is clamped at the maximum speed, regardless of whether the G96 mode or G97 mode is specified.

Alarm and message

Number	Message	Description
PS0190	ILLEGAL AXIS SELECTED (G96)	An illegal value was specified in P in a G96 block or parameter No. 3770.

Caution

⚠ CAUTION

1 If the spindle speed corresponding to the calculated surface speed exceeds the speed specified in the spindle speed clamp command (G50S_ for T series and G92S_ for M series) during the G96 mode, the actual spindle speed is clamped at the value specified in the spindle speed clamp command.
If the specified spindle speed is lower than the value specified in parameter No. 3771, the actual spindle speed is clamped at the speed specified by No. 3771.

2 When the constant surface speed control option is added to the M series (bit 0 (SSC) of parameter No. 8133 is set to 1), connections related to gear shift change also under normal spindle control.
(See "SPINDLE SPEED CONTROL".)

Note

NOTE

Simultaneous use of multi-spindle control enables constant surface speed control for spindles other than the first spindle. (See "MULTI-SPINDLE CONTROL.")

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Constant surface speed control (G96, G97)

10.9 ACTUAL SPINDLE SPEED OUTPUT (T SERIES)

T

Overview

The PMC can read actual spindle speed.

Signal

Actual spindle speed signals AR00 to AR15<Fn040,Fn041> : First spindle
AR002 to AR152<Fn202,Fn203> : Second spindle

[Classification] Output signal

[Function] These 16-bit binary code signals output from the CNC to the PMC the actual spindle speed obtained by feedback pulses from the position coder mounted on the spindle.

[Output cond.] These 16-bit binary code signals are always output. The actual spindle speed and the signals have the following relationship:

$$Spindle\ speed = \sum_{i=0}^{15} \left\{ 2^i \times V_i \right\} \min^{-1}$$

where Vi = 0 when ARi is "0" and Vi = 1 when ARi is "1"

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn040	AR07	AR06	AR05	AR04	AR03	AR02	AR01	AR00
Fn041	AR15	AR14	AR13	AR12	AR11	AR10	AR09	AR08
Fn202	AR072	AR062	AR052	AR042	AR032	AR022	AR012	AR002
Fn203	AR152	AR142	AR132	AR122	AR112	AR102	AR092	AR082

Note

NOTE
An absolute error of about 0.5 min^{-1} exists as a measuring error.
If the speed exceeds 65535min^{-1}, 65535 is assumed and output.

10.10 SPINDLE POSITIONING (T SERIES)

T

Overview

This function positions the spindle using the spindle motor and position coder.
The function has a coarser least command increment compared with the Cs contour control function and has no interpolation capability with other axes. However, it can be installed more easily because the position detector is a position coder.
Generally, the spindle positioning axes are clamped mechanically except when positioning is under way.

NOTE
1 Be sure to set 1 in bit 1 (AXC) of parameter No.8133 and 0 in bit 2 (SCS) of parameter No.8133 to use the spindle positioning function.
2 Both serial spindle Cs contour control function and spindle positioning function cannot be made effective at the same time.
If both are specified as AXC=1 and SCS=1, both functions become invalid.
Therefore, when a negative value is set in parameter No.1023 with above specification, alarm (SV1026) is generated.
3 Be sure to set 0 in bit 1 (AXC) of parameter No.8133 and 1 in bit 2 (SCS) of parameter No.8133 to use the serial spindle Cs contour control function.

Explanation

In the turning process, the workpiece is rotated by the spindle to which it is attached (spindle rotation mode), at the speed specified for the spindle motor. This spindle control state is called the spindle rotation mode.
The workpiece mounted on the spindle can be positioned at a certain angle by moving the spindle attached to the spindle motor by the certain angle. This function is called the spindle positioning function. This spindle control state is called the spindle positioning mode.

The spindle positioning function can perform the following operations:

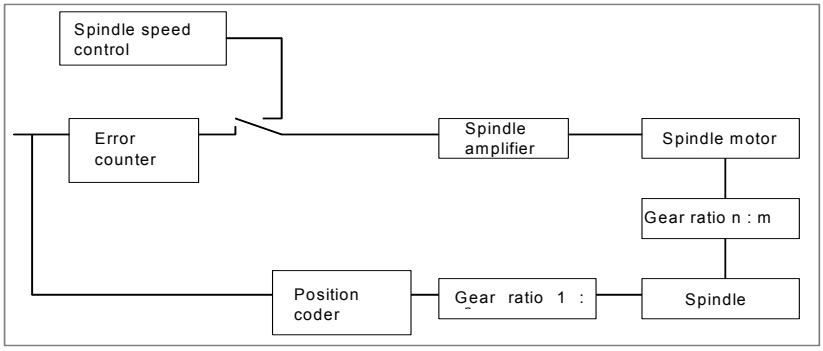
- Release the spindle rotation mode and enter the spindle positioning mode
  Specifying a particular M code sets a reference position in the spindle positioning mode. (spindle orientation)
- Position the spindle in the spindle positioning mode
  1) Position an optional angle using axis address,
  2) Position a semi-fixed angle using a specific M code parameter.

- Release the spindle positioning mode and enter the spindle rotation mode
  Specifying a particular M code parameter changes the spindle to the spindle rotation mode.

The least command increment, least input increment, and maximum value for the spindle positioning axis are as follows:

- Least command increment
  360/4096 ≒ 0.088 deg (When the gear ratio of the spindle to position coder is 1:1)
- Least input increment
  0.001 deg (IS-B)
- Max. programmable dimension
  ±999999.999 deg

- Control block diagram



The spindle positioning function is enabled only when the number of pulses of the position coder is 4096 pulses and the gear ratio of the spindle to position coder is one of the ratios listed below.

$1:2^n$ (n: Integer not smaller than 0)

The least command increment (detection unit) for each gear ratio is as follows:

Gear ratio of spindle to position coder	Least command increment (detection unit) deg
1 : 1	0.088 (1×360/4096)
1 : 2	0.176 (2×360/4096)
1 : 4	0.352 (4×360/4096)
1 : 8	0.703 (8×360/4096)
⋮	⋮
1 : N	(N×360/4096)

- Selecting a spindle positioning axis

As the target axis for spindle positioning, any axis address can be set in parameter No. 1020.
When setting the servo axis number of a spindle positioning axis (in parameter No. 1023), add a minus sign (-) to the spindle number for which spindle positioning is to be performed.
Spindle positioning axes are assigned after the end of controlled axes.

Setting example 1)
Servo axis : X, Z
Spindle positioning axis : C (first spindle), B (second spindle)
Number of spindles : 2

Number of controlled axes	Parameter No. 1020	Parameter No. 1023
1	88(X)	1
2	90(Z)	2
3	67(C)	-1
4	66(B)	-2

- Switching to spindle positioning mode

Switching to the spindle positioning mode is specified using the M code set in parameter No. 4960.
When the spindle motor is used for the ordinary spindle operation (the spindle rotation mode), switching to the spindle positioning mode must be made before spindle positioning can be performed.

Spindle orientation

After switching to the spindle positioning mode is completed, a spindle orientation is performed. The orientation function stops the spindle at a predetermined position.

To specify orientation, use the M code specified in parameter No. 4960. The orientation direction is specified by using parameters ZMI No. 1006 #5 for analog or RETSV No. 4000#4 for serial spindles.
With the grid shift function, the orientation position can be shifted within ±180 degrees by setting parameter No. 1850 when the spindle is an analog spindle, or can be shifted within the range from 0 to 360 degrees by setting parameter No. 4073 when the spindle is a serial spindle.

Orientation speed

When an analog spindle is used, a movement is made at the rapid traverse rate used for reference position return, which is set in parameter No. 1428, until the orientation enable speed is attained, and after the one-rotation signal is detected, orientation is performed at the FL feedrate set in parameter No. 1425.

* Orientation enable speed

$$1024 \geq \left(\frac{F \times P}{60 \times G \times 0.088} \right) \geq 128\,(pulses)$$

Set the orientation enable speed so that the expression shown is satisfied:
F : Rapid traverse rate for reference position return (deg/min)
G : Position gain (1/sec)
P : Gear ratio of the spindle to position coder

If the above condition is not satisfied, the one-rotation signal cannot be detected, so the spindle continues moving at the rapid traverse rate.
The rapid traverse rate for reference position return cannot be overridden.

* FL speed

Set the FL feedrate so that the following expression is satisfied:

$$1024 \geq \left(\frac{FL \times P}{60 \times G \times 0.088} \right) \geq effective\,area\,of\,spindle\,(pulses)$$

FL : Feedrate (deg/min)
G : Position gain (1/sec)
P : Gear ratio of the spindle to position coder

If the above condition is not satisfied, the accurate stop position cannot be detected, so the orientation is started again from the rapid traverse operation.

When a serial spindle is used, the orientation speed is determined by parameter settings on the spindle side.
When an orientation is performed for a serial spindle, the spindle motor rotates several turns, then the spindle is stopped at the orientation position.

Omitting orientation

When the orientation at the time of switching to the spindle positioning mode is not required (for example, when no start position is specified, and only incremental positioning from the current position is needed), the orientation performed at the time of switching to the spindle positioning mode can be omitted by setting bit 2 (ISZ) of parameter No. 4950. In this case, when the M code for switching to the spindle positioning mode is specified, the spindle control mode is just changed to the spindle positioning mode without orientation. This completes the switching to the spindle positioning mode.

Program origin

The position at which the orientation is completed is regarded as the program origin. The program origin can be changed by coordinate system setting (G92 (T series G code B, C/M series) or G50 (T series G code A)) or automatic coordinate system setting (bit 0 (ZPR) of parameter No. 1201).
When the orientation is omitted, the program origin is not established, so operations by absolute commands in spindle positioning by axis address are not guaranteed.

- Command system

The command system comes in two types: The first positions a semi-fixed angle; the second positions an optional angle.

Semi-fixed angle positioning

Specify a positioning angle with an M code. Six types of M codes Mα to M(α + 5) can be specified. The value of α is set in parameter No. 4962 in advance. The positioning angles corresponding to Mα to M(α + 5) are listed below, and β is set in parameter No. 4963 in advance.

M code (Example) β=α+5	Indexing angle	(Example) 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 (γ) of M codes used is specified in parameter No. 4964, M codes Mα to M(α + (γ - 1)) can be specified in up to 255 ways (Mα to M(α + (255 - 1))).

M code (Example) γ = 11	Indexing angle	(Example) β=30°
Mα	β	30°
M (α+1)	2β	60°
M (α+2)	3β	90°
M (α+3)	4β	120°
to	to	to
M (α+11-1)	11β	330°

One of the rotation directions can be set by setting bit 1 (IDM) of parameter No. 4950.

Optional angle positioning

The position at any angle (address H in an incremental command for the C-axis) is specified by using an axis address followed by a signed numeric value. The axis address must be specified in the G00 mode.
(In the subsequent explanation, the axis address is assumed to be the C-axis.)
(Example)

C-45000
C180.000

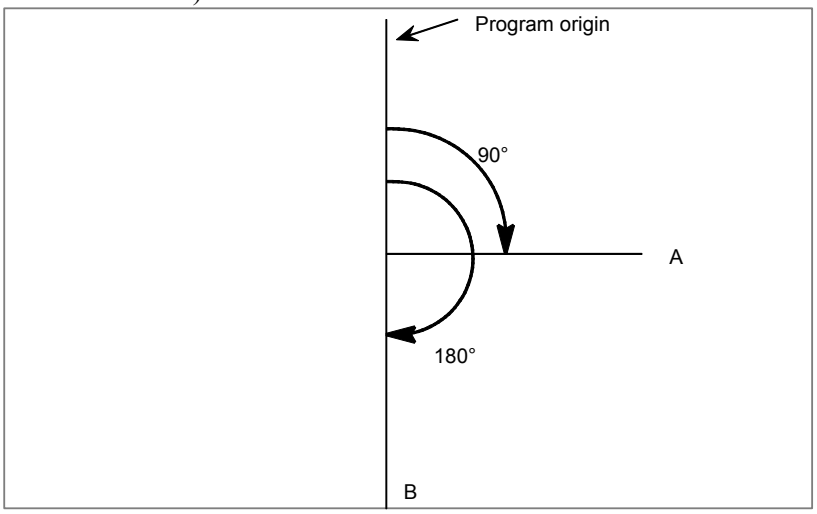
A numeric value with a decimal point can be entered. The position of the decimal point denotes the degree position.
(Example)

C36.0 = C36

Absolute and incremental commands

Positioning by specifying a semi-fixed angle (by M code) is always incremental.
One of the rotation directions can be set by setting bit 1 (IDM) of parameter No. 4950.

For positioning at any angle, an absolute command and an incremental command can be specified.
When an absolute command is used for positioning at any angle, short-cut control (by setting bit 1 (RAB) of parameter No. 1008 to 1) is also enabled using the rotation axis rollover function (by setting bit 0 (ROA) of parameter No. 1008 to 1).



Command method		G code system A for T series		G code system B, C for T series	
		Address used	Command of A-B on the above Figure	Address used and G-code	Command of A-B on the above Figure
Absolute command	Direct the end point position by the distance from the program	C	C180.0 ;	G90, C	G90C180.0 ;
Incremental command	Command by the distance between the start and end points.	H	H90.0 ;	G91, C	G91C90.0 ;

Spindle positioning feedrate

Spindle positioning is done at the rapid traverse rate specified by parameter No. 1420.
Overrides of 100%, 50%, 25% and F0 (parameter No. 1421) are also applied.

- Spindle positioning reset

A specific M code (parameter No. 4961) must be set when the mode is changed from spindle positioning to normal spindle rotation.

Also in the following cases, the spindle positioning mode is canceled, and the spindle rotation mode is entered:

<1> A reset operation (including emergency stop) is performed when a servo alarm is issued.
<2> A reset operation (including emergency stop) is performed when a spindle alarm is issued.
<3> The orientation operation is stopped in the middle by a reset, alarm, and so on.
<4> A reset operation (including emergency stop) is performed when bit 0 (IOR) of parameter No. 4950 is set to 1.

Diagnosis display

1544	Spindle positioning sequence status

The spindle positioning sequence status is indicated. (For each spindle)

1) Sequence for switching to the spindle positioning mode

Indication	Description
00010003	Wait until SPSTPs is turned on (0->1)
00010005	Wait for completion of switching to spindle positioning mode
00010006	Wait until *SUCPFs is turned on (1->0)
00010008	Wait for completion of orientation operation
00010009	Wait for completion of preset processing after orientation
0001000A	Wait until *SCLPFs is turned on (1->0)

2) Spindle positioning sequence

Indication	Description
00020004	Wait until SPSTPs is turned on (0->1) Wait until *SUCPFs is turned on (1->0)
00020005	Movement in progress
00020006	Wait for completion of positioning operation
00020007	Wait until *SCLPFs is turned on (1->0)

For spindle positioning sequence B specifications
(when bit 7 (IMB) of parameter No. 4950 is set to 1)

00040003	Wait until SPSTPs is turned on (0->1)
00040005	Wait for completion of switching to spindle positioning mode
00040006	Wait until *SUCPFs is turned on (1->0)
00040008	Wait for completion of positioning operation
00040009	Wait until *SCLPFs is turned on (1->0)
0004000A	Wait for completion of spindle positioning mode cancellation

3) Spindle positioning mode cancellation sequence

Indication	Description
00030003	Wait until SPSTPs is turned on (0->1) Wait until *SUCPFs is turned on (1->0)
00030004	Wait for completion of spindle positioning mode cancellation
FFFFFFFF	End of sequence

5207	Clamp/unclamp sequence status

The clamp/unclamp sequence status is indicated. (For each controlled axis)

1) Sequence for switching to the spindle positioning mode

Indication	Description
00030001	Wait until SPSTPs is turned on (0->1)
00030002	Wait for completion of switching to spindle positioning mode
00030004	Wait until *SUCPFs is turned on (1->0)
00030005	Wait for completion of orientation operation
00030006	Wait until *SCLPFs is turned on (1->0)

2) Spindle positioning sequence

Indication	Description
00070001	Wait until SPSTPs is turned on (0->1)

00070002 Wait until *SUCPFs is turned on (1->0)
00070003 Wait for completion of positioning operation
00070004 Wait until *SCLPFs is turned on (1->0)

For spindle positioning sequence B specifications (when bit 7 (IMB) of parameter No. 4950 is set to 1)
000D0001 Wait until SPSTPs is turned on (0->1)
000D0002 Wait for completion of switching to spindle positioning mode
000D0004 Wait until *SUCPFs is turned on (1->0)
000D0005 Wait for completion of positioning operation
000D0006 Wait until *SCLPFs is turned on (1->0)
000D0007 Wait for completion of spindle positioning mode cancellation

3) Spindle positioning mode cancellation sequence
Indication Description
000B0001 Wait until SPSTPs is turned on (0->1)
000B0002 Wait until *SUCPFs is turned on (1->0)
000B0004 Wait for completion of spindle positioning mode cancellation

FFFFFFFF End of sequence

Spindle screen

In the spindle positioning mode, indications related to the spindle positioning mode appear.

Signal

Spindle unclamp signals

SUCLPA<Fn038.1> : First spindle
SUCLPB<Fn400.1> : Second spindle

[Classification] Output signal
[Function] These signals specify that spindle mechanical clamping be released in a spindle positioning sequence.
When this signal turns to 1, unclamp the spindle on the machine (release the brakes or extract the pin).
[Output cond.] Refer to the sequence (time chart) in this section.

Spindle unclamp completion signals

*SUCPFA<Gn028.4> : First spindle
*SUCPFB<Gn400.1> : Second spindle

[Classification] Input signal
[Function] These signals indicate that unclamping the spindle is complete in response to the spindle unclamp signal SUCLPs.

Spindle clamp signals

SCLPA<Fn038.0> : First spindle
SCLPB<Fn401.1> : Second spindle

[Classification] Output signal
[Function] These signals specify that the spindle be clamped mechanically in a spindle positioning sequence.
When this signal turns to 1, clamp the spindle on the machine (apply the brakes or insert the pin).
[Output cond.] Refer to the sequence (time chart) in this section.

Spindle clamp completion signals
*SCPFA<Gn028.5> : First spindle
*SCPFB<Gn401.1> : Second spindle

[Classification] Input signal

[Function] These signals indicate that clamping the spindle is complete in response to the spindle clamp signal SCLPs.

Spindle positioning mode signals
MSPOSA<Fn039.0> : First spindle
MSPOSB<Fn402.1> : Second spindle

[Classification] Output signal

[Function] These signals indicate that the spindle is in the spindle positioning mode.

[Output cond.] Refer to the sequence (time chart) in this section.

Spindle stop complete signal
SPSTPA<Gn028.6> : First spindle
SPSTPB<Gn402.1> : Second spindle

[Classification] Input signal

[Function] After checking that these signals are set to 1, the CNC performs spindle orientation, spindle positioning, and spindle positioning cancellation.

- Signals related to gear change

Gear selection signals GR1,GR2<Gn028.1, 2>

[Classification] Input signal

[Function] These signals inform the CNC of the gear stage currently selected.

[Operation] For information about these signals, see "SPINDLE SPEED CONTROL".

Clutch/gear signals (serial spindle)
CTH1A,CTH2A<Gn070.3,Gn070.2> : First spindle
CTH1B,CTH2B<Gn074.3,Gn074.2> : Second spindle

See the relevant manual of the serial spindle.

These signals determine the gear parameters (such as the loop gain) to be used.
Although the gear selection signals for a serial spindle are CTH1 and CTH2, also input GR1 and GR2. Avoid changing the states of these signals in the spindle positioning mode.

Relationship between selected gear and spindle gear selection signals

CNC side T series			Serial spindle side	
GR2	GR1	Selected gear	CTH1	CTH2
0	0	1st gear	0	0
0	1	2nd gear	0	1
1	0	3rd gear	1	0
1	1	4th gear	1	1

NOTE
The above combinations of clutch/gear signals (CTH1 and CTH2) are just examples.
Since the serial spindle selects a gear by using CTH1 and CTH2 independent of gear selection on the CNC, the signals must be input as necessary, and serial spindle parameters must be set accordingly.

- Other signals

Reference position return completion signal ZPx<Fn094>

[Classification] Output signal

[Function] This signal indicates that spindle orientation of the spindle positioning axis is completed.

[Output cond.] This signal is set to 1 when spindle orientation is completed, and the signal is set to 0 when spindle positioning or spindle positioning cancellation is performed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn028		SPSTPA	*SCPFA	*SUCPFA				
Gn028						GR2	GR1	
Gn400							*SUCPFB	
Gn401							*SCPFB	
Gn402							SPSTPB	
Fn038							SUCLPA	SCLPA
Fn039								MSPOSA
Fn094				ZP5	ZP4	ZP3	ZP2	ZP1
Fn400							SUCLPB	
Fn401							SCLPB	
Fn402							MSPOSB	

- For first serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn070					CTH1A	CTH2A		

- For second serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn074					CTH1B	CTH2B		

- Sequence (Time chart)

Switching to the spindle positioning mode

The operation of canceling the spindle rotation mode to enter the spindle positioning mode is performed by specifying an M code for switching to the spindle positioning mode.

Set the M code for switching to the spindle positioning mode in parameter No. 4960 in advance.

<1> Suppose that an M code is specified by a program.

<2> Auxiliary function code signal M00 to M31 is sent, and the auxiliary function strobe signal MF is set to 1 at the same time. This sending procedure is the same as for ordinary auxiliary functions.

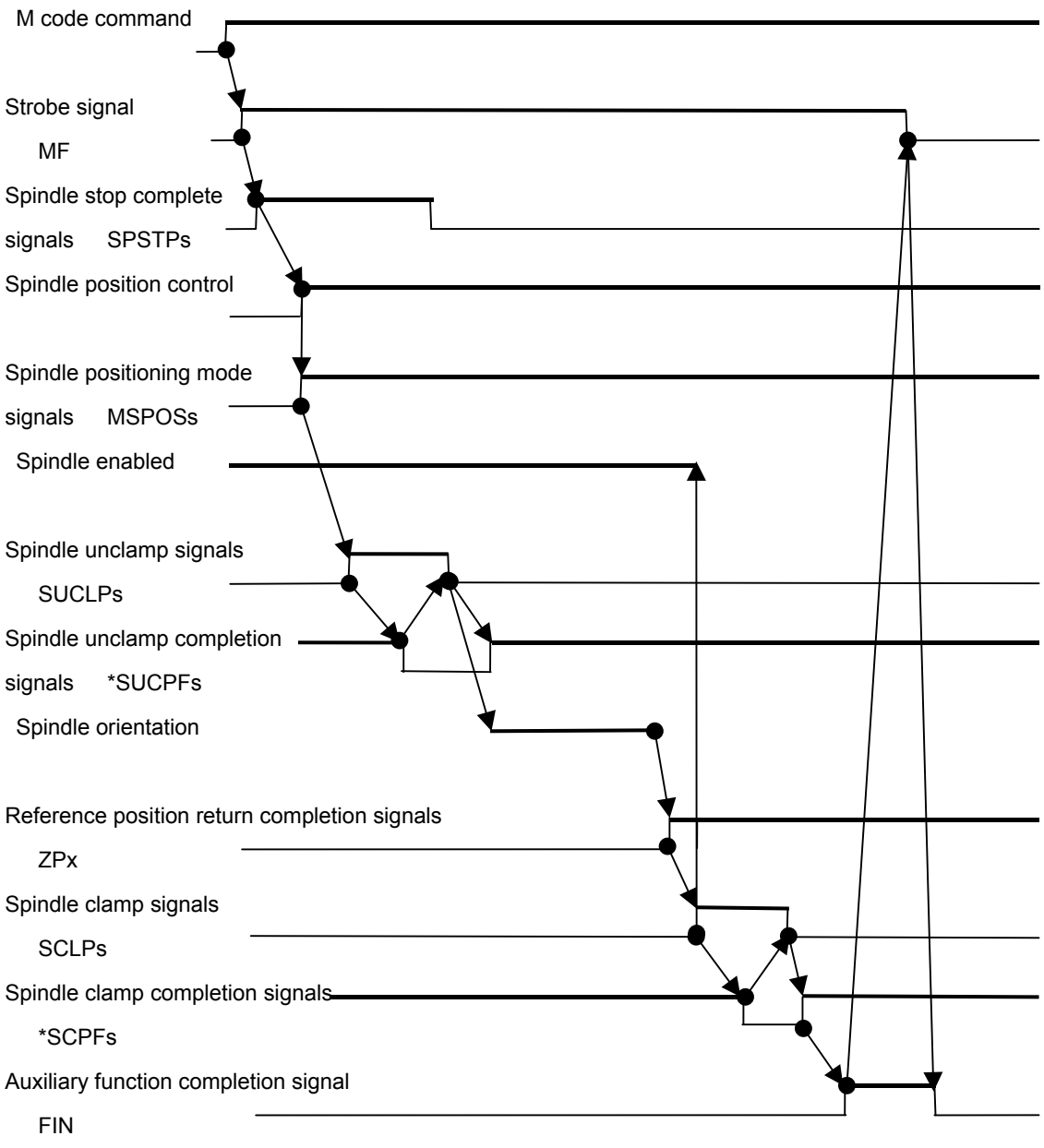
<3> When strobe signal MF is set to 1, the PMC must read the code signal and perform corresponding processing. After completing the corresponding processing, the PMC sets spindle stop complete signal SPSTPs to 1 when the spindle is stopped.

<4> As soon as setting strobe signal MF to 1, the CNC checks whether spindle stop complete signal SPSTPs is set to 1. When SPSTPs is set to 1, the CNC changes the spindle control mode from the spindle rotation mode to spindle positioning mode, and sets spindle positioning mode signal MSPOSs to 1.

<5> The CNC sets spindle unclamp signal SUCLPs to 1.

<6> When spindle unclamp signal SUCLPs is set to 1, the PMC performs processing to make the spindle usable for positioning. When completing the processing, the PMC sets spindle unclamp completion signal *SUCPFs to 0.
<7> When spindle unclamp completion signal *SUCPFs are set to 0, the CNC sets spindle unclamp signal SUCLPs to 0 to post that the CNC has received the *SUCPFs signal.
<8> When spindle unclamp signal SUCLPs is set to 0, the PMC must set spindle unclamp completion signal *SUCPFs to 1.
<9> The CNC rotates the spindle and stops the spindle at a certain position. (This is generally called spindle orientation.) Then, the CNC sets reference position return completion signal ZPx (x denotes the controlled axis number of the spindle positioning axis) to 1.
<10>The CNC sets spindle clamp signal SCLPs to 1.
<11>When spindle clamp signal SCLPs is set to 1, the PMC clamps the spindle mechanically with a device such as a clutch or shot pin as necessary. After completing the clamping operation, the PMC sets spindle clamp completion signal *SCPFs to 0.
<12>When spindle clamp completion signal *SCPFs is set to 0, the CNC sets spindle clamp signal SCLPs to 0 to post that the CNC has received *SCPFs.
<13>When spindle clamp signal SCLPs is set to 0, the PMC sets spindle clamp completion signal *SCPFs to 1.
<14>Finally, the PMC sets completion signal FIN to 1 for auxiliary function strobe signal MF. In response to this, the CNC sets the MF signal to 0. Then, the PMC sets the FIN signal to 0.

Timing chart for switching to spindle positioning mode



⚠ CAUTION

1 Switching between the spindle rotation mode and spindle positioning mode must be performed on the machine side in response to M code commands issued from the CNC.
When the spindle positioning mode is canceled, the spindle becomes ready for accepting speed commands such as an S command. Therefore, a PMC ladder program must be created so that spindle rotation is not performed when the spindle is clamped on the machine side.
Example:
When the spindle is clamped on the machine side, spindle stop signal *SSTP <G029.6> is set to 0.

⚠ CAUTION

2 When the spindle is clamped on the machine side (by using a device such as a clutch or shot pin), the spindle must be disabled. Enabling or disabling the spindle means to separately instruct the spindle control unit to enable or disable spindle motor operation by the PMC ladder and so on.
Example:
When a serial spindle is enabled, SFRA <G070.5> is set to 1. When the serial spindle is disabled, SFRA <G070.5> is set to 0.
Refer to the manual of the spindle control unit to be connected, and add instructions to the ladder program.

Spindle positioning

In the spindle positioning mode, positioning is performed using one of the following two programmed commands:

1) Positioning at a semi-fixed angle by specifying an M code
2) Positioning at any angle by specifying an axis address

The operations performed by using the above two specification methods are the same, except that in positioning at any angle, input/output signals related to auxiliary functions are not exchanged.
The M code used for positioning at a semi-fixed angle is set in parameter Nos. 4962 and 4964 in advance.

When positioning at a semi-fixed angle is performed by specifying the M code, specification A or B can be selected by setting bit 7 (IMB) of parameter No. 4950.

If specification B is selected, specifying an M code that specifies the spindle positioning angle performs operations (1) to (3) below successively.

(1) Canceling the spindle rotation mode and switching to the spindle positioning mode
(2) Positioning the spindle in the spindle positioning mode
(3) Canceling the spindle positioning mode and switching to the spindle rotation mode

If specification A is selected, only the spindle positioning operation is performed in the spindle positioning mode.

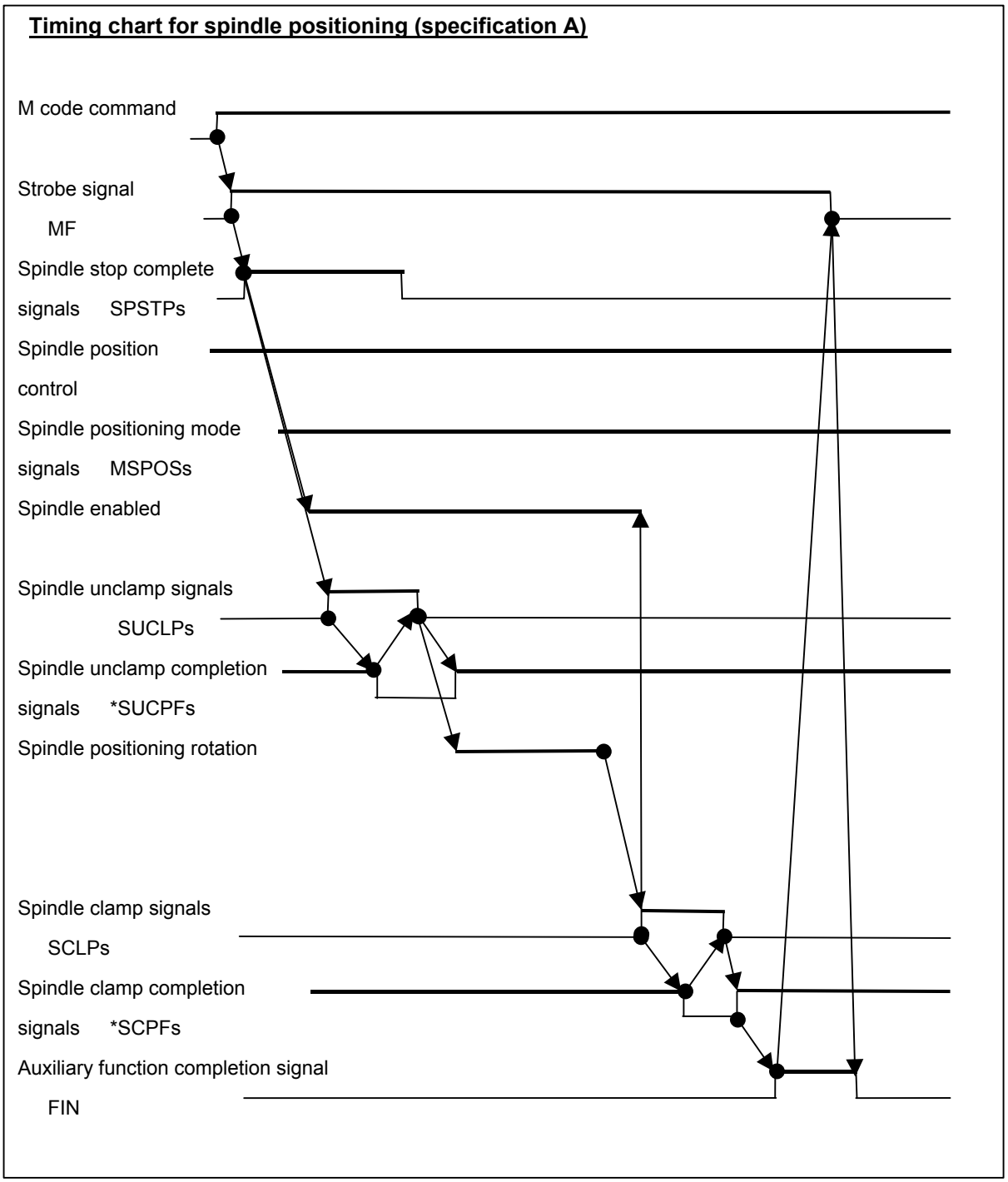
[Specification A: Bit 7 (IMB) of parameter No. 4950 = 0]

<1> Suppose that an M code is specified by a program. (Note)
<2> Auxiliary function code signal M00 to M31 is sent, and the auxiliary function strobe signal MF is set to 1 at the same time. This sending procedure is the same as for ordinary auxiliary functions.
<3> When strobe signal MF is set to 1, the PMC must read the code signal and perform corresponding processing. After completing the corresponding processing, the PMC sets spindle stop complete signal SPSTPs to 1 when the spindle is stopped.
<4> As soon as setting strobe signal MF to 1, the CNC checks whether spindle stop complete signal SPSTPs is set to 1. When SPSTPs is set to 1, the CNC sets the spindle unclamp signal SUCLPs to 1.
<5> When spindle unclamp signal SUCLPs is set to 1, the PMC unclamps the spindle, and when unclamping of the spindle is completed, the PMC sets spindle unclamp completion signal *SUCPFs to 0.
<6> When spindle unclamp completion signal *SUCPFs is set to 0, the CNC sets spindle unclamp signal SUCLPs to 0 to post that the CNC has received the *SUCPFs signal.
<7> When spindle unclamp signal SUCLPs is set to 0, the PMC must set spindle unclamp completion signal *SUCPFs to 1. After setting the SUCLPs signal to 0, the CNC rotates the spindle, makes a movement as specified, then stops the spindle.
<8> The CNC sets spindle clamp signal SCLPs to 1.
<9> When spindle clamp signal SCLPs is set to 1, the PMC clamps the spindle mechanically with a device such as a clutch or shot pin as necessary. After completing the clamping operation, the PMC sets spindle clamp completion signal *SCPFs to 0.

<10>When spindle clamp completion signal *SCPFs is set to 0, the CNC sets spindle clamp signal SCLPs to 0 to post that the CNC has received *SCPFs.
<11>When spindle clamp signal SCLPs is set to 0, the PMC sets spindle clamp completion signal *SCPFs to 1.
<12>Finally, the PMC sets completion signal FIN to 1 for auxiliary function strobe signal MF. In response to this, the CNC sets the MF signal to 0. Then, the PMC sets the FIN signal to 0.

NOTE
If an M code is specified in the spindle rotation mode, the CNC issues alarm PS0224.

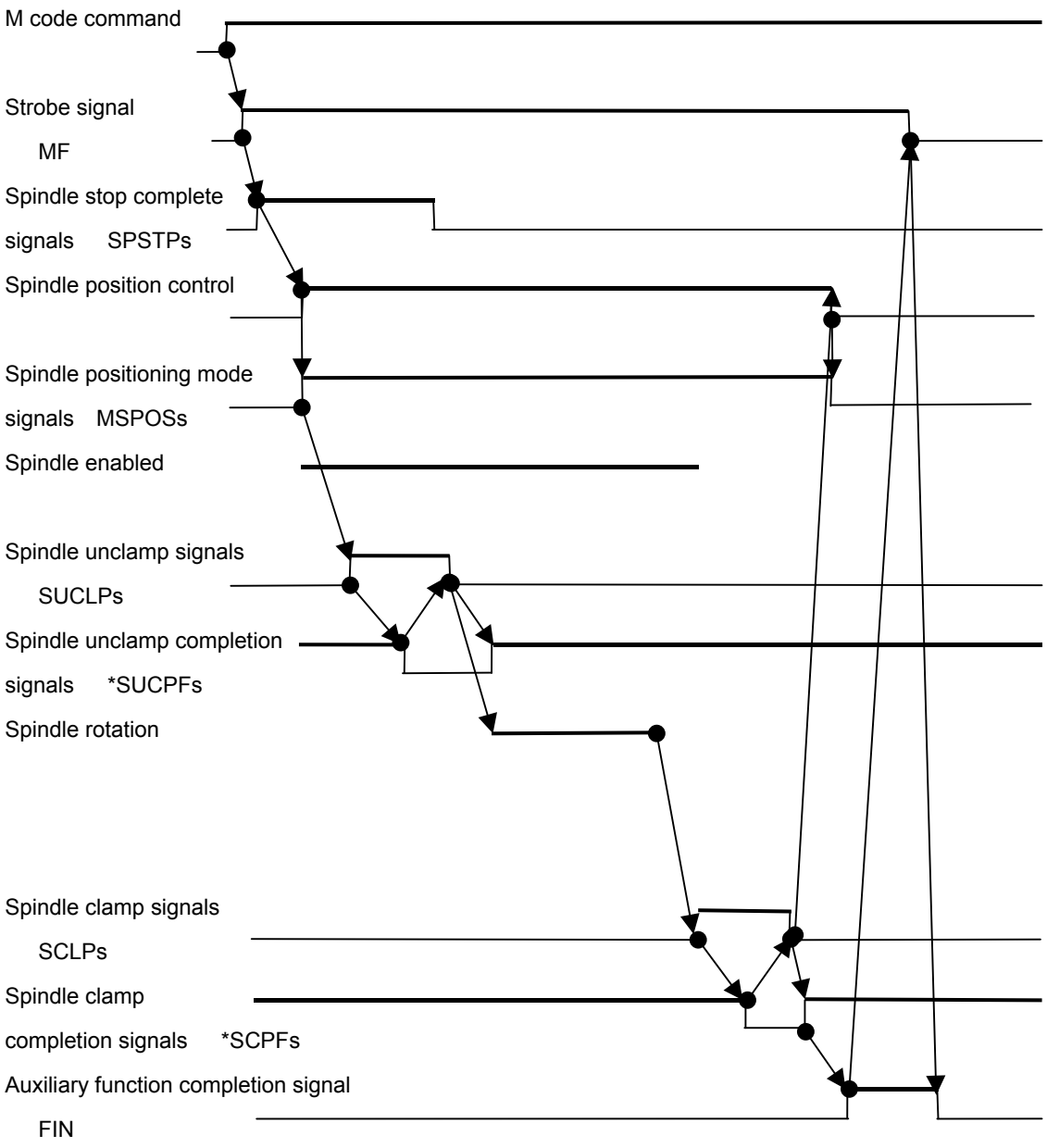
Timing chart for spindle positioning (specification A)



[Specification B: Bit 7 (IMB) of parameter No. 4950 = 1]

<1> Suppose that an M code is specified in a program.
<2> Auxiliary function code signal M00 to M31 is sent, and the auxiliary function strobe signal MF is set to 1 at the same time. This sending procedure is the same as for ordinary auxiliary functions.
<3> When strobe signal MF is set to 1, the PMC must read the code signal and perform corresponding processing. After completing the corresponding processing, the PMC sets spindle stop complete signal SPSTPs to 1 when the spindle is stopped.
<4> As soon as setting strobe signal MF to 1, the CNC checks whether spindle stop complete signal SPSTPs is set to 1. When the SPSTPs signal is set to 1, the CNC switches the spindle control mode from the spindle rotation mode to spindle positioning mode, and sets spindle positioning mode signal MSPOSs to 1.
<5> The CNC sets spindle unclamp signal SUCLPs to 1.
<6> When spindle unclamp signal SUCLPs is set to 1, the PMC unclamps the spindle, and when unclamping of the spindle is completed, the PMC sets spindle unclamp completion signal *SUCPFs to 0.
<7> When spindle unclamp completion signal *SUCPFs is set to 0, the CNC sets spindle unclamp signal SUCLPs to 0 to post that the CNC has received the *SUCPFs signal.
<8> When spindle unclamp signal SUCLPs is set to 0, the PMC sets spindle unclamp completion signal *SUCPFs to 1.
<9> After setting the SUCLPs signal to 0, the CNC rotates the spindle, makes a movement as specified, then stops the spindle.
<10>The CNC sets spindle clamp signal SCLPs to 1.
<11>When spindle clamp signal SCLPs is set to 1, the PMC clamps the spindle mechanically with a device such as a clutch or shot pin as necessary. After completing the clamping operation, the PMC sets spindle clamp completion signal *SCPFs to 0.
<12>When spindle clamp completion signal *SCPFs is set to 0, the CNC sets spindle clamp signal SCLPs to 0 to post that the CNC has received *SCPFs. The CNC sets the SCLPs signal to 0, and switches the spindle control mode from the spindle positioning mode to spindle rotation mode at the same time. The CNC sets spindle positioning mode signal MSPOSs to 0.
<13>When spindle clamp signal SCLPs is set to 0, the PMC sets spindle clamp completion signal *SCPFs to 1.
<14>Finally, the PMC sets completion signal FIN to 1 for auxiliary function strobe signal MF. In response to this, the CNC sets the MF signal to 0. Then, the PMC sets the FIN signal to 0.

Timing chart for spindle positioning (specification B)



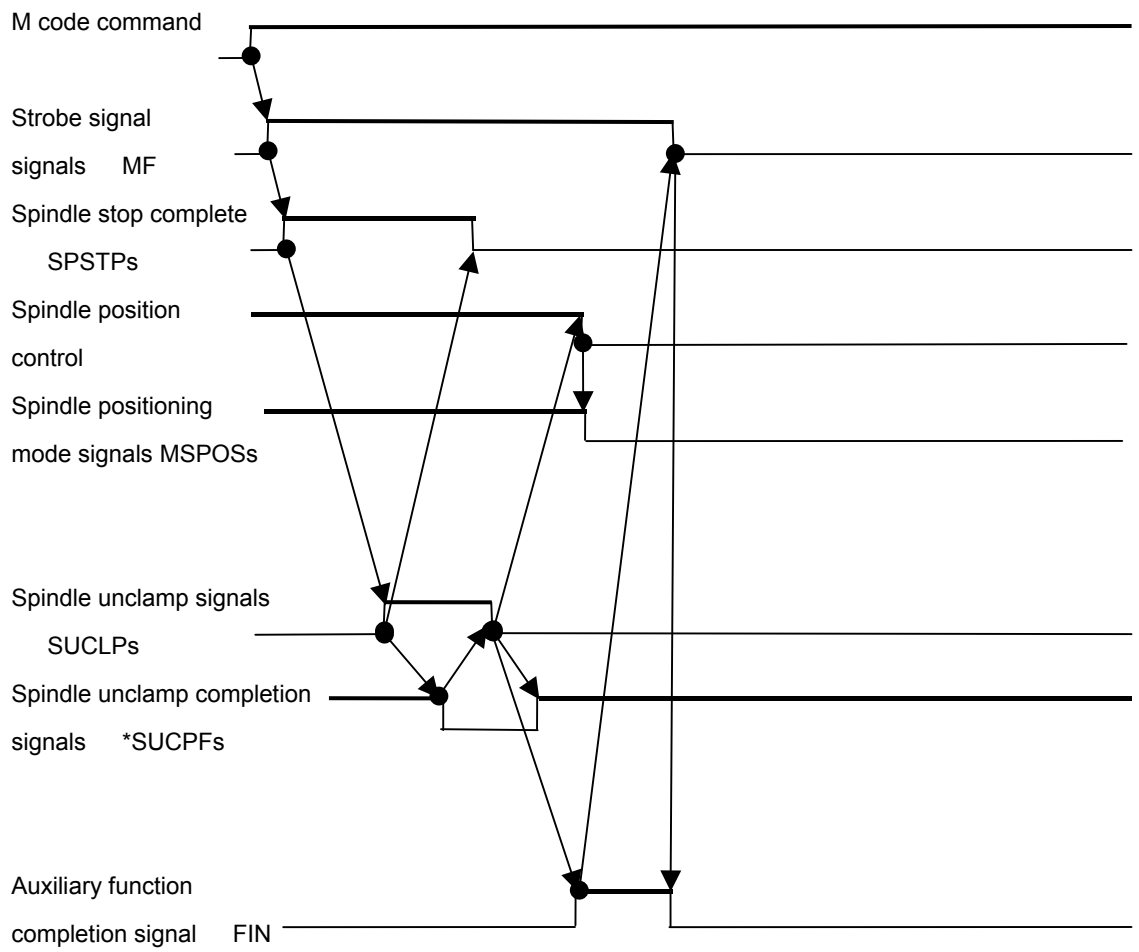
Canceling spindle positioning

To cancel the spindle positioning mode and enter the spindle rotation mode again, use an M code for canceling the spindle positioning mode.
Set the M code in parameter No. 4961 in advance.

<1> Suppose that an M code is specified in a program.
<2> Auxiliary function code signal M00 to M31 is sent, and the auxiliary function strobe signal MF is set to 1 at the same time. This sending procedure is the same as for ordinary auxiliary functions.
<3> When strobe signal MF is set to 1, the PMC must read the code signal and perform corresponding processing. After completing the corresponding processing, the PMC sets spindle stop complete signal SPSTPs to 1 when the spindle is stopped.

<4> As soon as setting strobe signal MF to 1, the CNC checks whether spindle stop complete signal SPSTPs is set to 1. When the SPSTPs signal is set to 1, the CNC sets spindle unclamp signal SUCLPs to 1.
<5> When spindle unclamp signal SUCLPs is set to 1, the PMC unclamps the spindle, and when unclamping of the spindle is completed, the PMC sets spindle unclamp completion signal *SUCPFs to 0.
<6> When spindle unclamp completion signal *SUCPFs is set to 0, the CNC sets spindle unclamp signal SUCLPs to 0 to post that the CNC has received the *SUCPFs signal. As soon as setting the SUCLPs signal to 0, the CNC switches the spindle control mode from the spindle positioning mode to spindle rotation mode. The CNC sets spindle positioning mode signal MSPOSs to 0.
<7> When spindle unclamp signal SUCLPs is set to 0, the PMC sets spindle unclamp completion signal *SUCPFs to 1.
<8> Finally, the PMC sets completion signal FIN to 1 for auxiliary function strobe signal MF. In response to this, the CNC sets the MF signal to 0. Then, the PMC sets the FIN signal to 0.

Timing chart for canceling spindle positioning mode



Parameter

Major related parameters are described below.

	#7	#6	#5	#4	#3	#2	#1	#0
1005								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
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.)

	#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 rotation axis.
#1 ROSx

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	Rotation 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 parameter No.1008#0(ROAx) and #2(RRLx). (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	Rotation axis (B type) (1) Inch/metric conversion, absolute coordinate values and relative coordinate values are 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 rotation 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

NOTE
For the FS0*i*-C, one of the following changes is required besides setting bit 3 (DIAx) of parameter No. 1006 so that the axis based on diameter specification achieves the specified amount of movement.
- Halve the command multiplication (the detection unit is not changed).
- Halve the detection unit and double the flexible feed gear (DMR).

For the FS0*i*-D, only if bit 3 (DIAx) of parameter No. 1006 is set, the CNC halves the specified pulse. Accordingly, the above changes are not required (when the detection unit is not changed). To halve the detection unit, double both CMR and DMR.

#5 ZMIx The direction of manual reference position return is:
0: + direction
1: - direction

	#7	#6	#5	#4	#3	#2	#1	#0
1013							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 ISA
#1 ISC Increment system of each axis

Increment system	#1 ISC	#0 ISA
IS-A	0	1
IS-B	0	0
IS-C	1	0

IS-B is specified for the spindle positioning axis. So, set ISA to ISC to 0.

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/too nose radius compensation, and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane, G19: Yp-Zp plane), 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	Rotation 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

Setting	Meaning
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 Number of controlled axes
Set the servo axis for each control axis.
Set the spindle number targeted for spindle positioning preceded by a minus sign (-1 to –2), as the servo axis number of an axis used as the spindle positioning axis.
Settings
-1 : Spindle positioning by using the first spindle
-2 : Spindle positioning by using the second spindle

NOTE
One spindle cannot be set as multiple spindle positioning axes.

1260 **Amount of a shift per one rotation of a rotation 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 amount of a shift per one rotation of a rotation axis.
For the rotation axis used for cylindrical interpolation, set the standard value.
For the spindle positioning axis, set 360.0.

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.

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

1420: rapid traverse rate
1423: Jog feedrate
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.

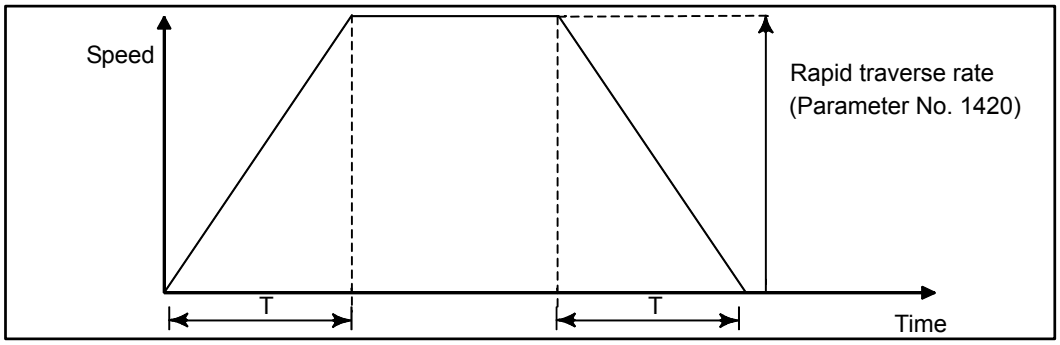
*3 : When rapid traverse is used for reference position return without dogs or manual reference position return after reference position establishment, regardless of the deceleration dog, the feedrate for manual reference position return based on these functions is used (the setting of bit 1 (DLF) of parameter No. 1404 is followed).

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

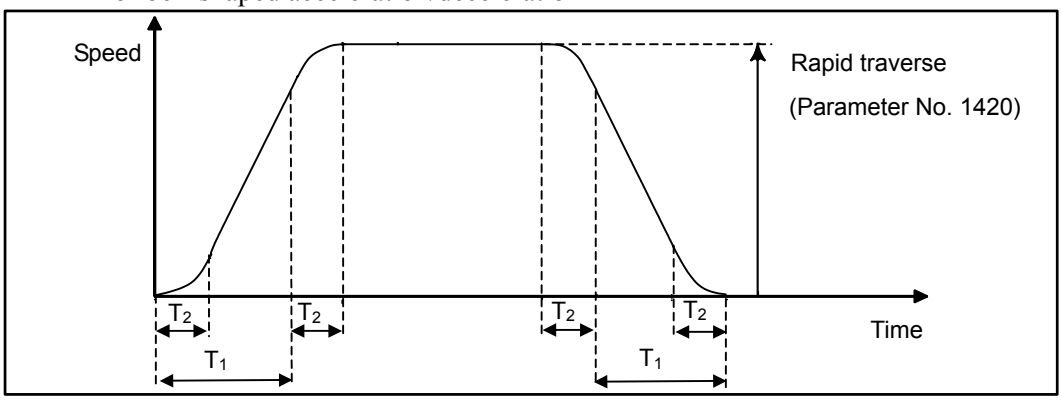
Specify a time constant used for acceleration/deceleration in rapid traverse.

[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 : $T_2 \times 2$

1621	Time constant T2 used for 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 1000
Specify time constant T2 used for bell-shaped acceleration/ deceleration in rapid traverse for each axis.

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] 1 to 96
Set 2 for spindle positioning.

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.

1826	In-position width for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
The in-position width is set for each axis.
When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in-position width, the machine is assumed to have reached the specified position. (The machine is in the in-position state.)

1827	In-position width in cutting feed for each axis

[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 each axis in cutting feed. This parameter is used when bit 4 (CCI) of parameter No.1801=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) 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) is generated, and operation is stopped immediately (as in emergency stop).

1850	Grid shift and reference position shift 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 99999999
To shift the reference position, the grid can be shifted by the amount set in this parameter. Up to the maximum value counted by the reference counter can be specified as the grid shift.
In case of parameter SFDx(No.1008#4) is 0: Grid shift
In case of parameter SFDx(No.1008#4) is 1: Reference point shift

NOTE
For setting the reference position without dogs, only the grid shift function can be used.
(The reference position shift function cannot be used.)

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.

3720	Number of position coder pulses

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] Detection unit
[Valid data range] 1 to 32767
Set the number of position coder pulses.

3721	Number of gear teeth on the position coder side

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 9999

Set the number of gear teeth on the position coder side on speed control (such as feed per revolution and threading).

3722	Number of gear teeth on the spindle side

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 9999
Set the number of gear teeth on the spindle side on speed control (such as feed per revolution and threading).

	#7	#6	#5	#4	#3	#2	#1	#0
4000				RETSV				

[Input type] Parameter input
[Data type] Bit spindle

#4 RETSV Reference position return direction of spindle on servo mode. (spindle positioning, rigid tapping, etc.)
0 : CCW (Counter clockwise)
1 : CW (Clockwise)

NOTE
The direction for spindle orientation (or reference position return) in spindle positioning using a serial spindle is determined by this parameter.

4056	Gear ratio (HIGH)
4057	Gear ration (MEDIUM HIGH)
4058	Gear ratio (MEDIUM LOW)
4059	Gear ratio (LOW)

[Data type] Word spindle
[Unit of data] Motor speed per spindle rotation × 100
[Valid data range] 0 to 32767
These parameters set the gear ration between the spindle and spindle motor.

NOTE
Set the gear ration between spindle and AC spindle motor when the spindle positioning is performed with serial spindle. For which gear is used, it depends on the clutch/gear signal (serial spindle) CTH1A, CTH1B.

4065	Position gain in servo mode (HIGH)
4066	Position gain in servo mode (MEDIUM HIGH)
4067	Position gain in servo mode (MEDIUM LOW)

4068	Position gain in servo mode (LOW)

[Data type] Word spindle
[Unit of data] 0.01 sec^{-1}
[Valid data range] 0 to 32767
These parameters set a servo loop gain on servo mode. (spindle positioning, rigid tapping, etc.)

NOTE
When the spindle positioning by a serial spindle is performed, set the position control loop gain in place of parameter No. 4970. For which gear is used, it depends on the clutch/gear signal (serial spindle) CTH1s, CTH1s.

4073	Grid shift amount on servo mode

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] 0to4096
This parameter sets a grid shift amount (distance from the position of a one-rotation signal to the machine zero point) in a servo mode (such as spindle positioning or rigid tapping).

NOTE
The setting of this parameter is used as a grid shift amount for orientation (reference position return) in spindle positioning using a serial spindle.

4074	Reference position return speed on servo mode

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] min^{-1}
The spindle speed set in this parameter is used for spindle orientation in a servo mode (such as spindle positioning or rigid tapping).

NOTE
1 The setting of this parameter is used as a grid shift amount for orientation (reference position return) in spindle positioning using a serial spindle.
2 If this parameter is set to 0, the reference position return feedrate is assumed to be the setting of parameter No. 4021.

	#7	#6	#5	#4	#3	#2	#1	#0
4950	IMBs	ESIs	TRVs			ISZs	IDMs	IORs

[Input type] Parameter input
[Data type] Bit spindle

#0 IORs Resetting the system in the spindle positioning mode
0: Does not releases 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 spindle orientation is specified in 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.)

#5 TRVs The rotation direction for spindle positioning is:
0: Same as the specified sign.
1: Opposite to the specified sign.

NOTE
When a serial spindle is used, this parameter is invalid for the specification of a rotation direction for the orientation command.

#6 ESIs The unit of rapid traverse rate on the spindle positioning axis is:
0: Not increased by a factor of 10.
1: Increased by a factor of 10.

#7 IMBs When the spindle positioning function is used, half-fixed angle positioning based on M codes uses:
0: Specification A
1: Specification B

In the case of half-fixed angle positioning based on M codes, three types of spindle positioning operations can occur:
(1) The spindle rotation mode is cleared, then the mode is switched to the spindle positioning mode. (After switching to the spindle positioning mode, spindle orientation operation is also performed.)
(2) Spindle positioning is performed in the spindle positioning mode.
(3) The spindle positioning mode is cleared, then the mode is switched to the spindle rotation mode.

- In the case of specification A:
  Operations (1) to (3) are specified using separate M codes.
  (1) Specified using an M code for switching to the spindle positioning mode. (See parameter No.4960)
  (2) Specified using M codes for specifying a spindle positioning angle. (See parameter No.4962)
  (3) Specified using M codes for clearing spindle positioning operation. (See parameter No.4961.)
- In the case of specification B:
  When M codes for specifying a spindle positioning angle are specified, operations (1) to (3) are performed successively. (See parameter No.4962.) (However, spindle orientation operation of (1) is not performed.)

	#7	#6	#5	#4	#3	#2	#1	#0
4959								DMDx

[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 DMDx A machine coordinate on the spindle positioning axis is displayed in:
0: Degrees.
1: Pulses.

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] 0.0 to 60.0
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).

4970	Position gain

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 0.01/sec
[Valid data range] 1 to 9999
Set the position gain of the analog spindle in the spindle positioning mode.

4971	Position gain multiplier (first stage)
to	
4974	Position gain multiplier (fourth stage)

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 1 to 32767
Set a position gain multiplier for an analog spindle in spindle positioning.
Position gain multiplier GC is obtained from the following equation:

$$GC = \frac{2048000 \times 360 \times PC \times E}{PLS \times SP \times L}$$

PLS Number of pulses output from the position coder (pulses/rev)
SP Number of gear teeth on the spindle side
PC Number of gear teeth on the position coder side
E Specified voltage (V) for turning the spindle motor at 1000 min^{-1}
L Angular displacement of the spindle (degrees) per spindle motor rotation
Example: For the spindle motor and gear ratio given below, GC is calculated as follows:

$$GC = \frac{2048000 \times 360 \times 1 \times 2.2}{4096 \times 1 \times 360} = 1100$$

PLS = 4096 pulse/rev
SP = 1
PC = 1
E = 2.2 V
L = 360 deg

NOTE
On the assumption that the spindle motor used turns at 4500 min^{-1} at 10 V, 2.2 V is required to turn the spindle motor at 1000 min^{-1}

Alarm and message

Number	Message	Description
PS0136	SPOS AXIS - OTHER AXIS SAME TIME	M code of the spindle positioning and the axis address which is not the spindle positioning axis are commanded at the same time. In the spindle positioning mode, the axis address of the spindle positioning axis and the axis address which is not the spindle positioning axis are commanded at the same time.
PS0137	M-CODE & MOVE CMD IN SAME BLK.	M code of the spindle positioning and the axis address which is the spindle positioning axis are commanded at the same time.
PS0194	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 simple spindle synchronous control mode.
PS0224	ZERO RETURN NOT FINISHED	1) A reference position 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. 2) The spindle positioning axis is commanded when the mode is not spindle positioning. Perform the spindle orientation.
PS1509	DUPLICATE M-CODE (SPOS AXIS ORIENTATION)	A function to which the same code as this M code is set exists. (spindle positioning, orientation)
PS1510	DUPLICATE M-CODE (SPOS AXIS POSITIONING)	A function to which the same code as this M code is set exists. (spindle positioning, positioning)
PS1511	DUPLICATE M-CODE (SPOS AXIS RELEASE)	A function to which the same code as this M code is set exists. (spindle positioning, mode cancel)
PS1543	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.
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.
SP1224	ILLEGAL SPINDLE-POSITION CODER GEAR RATIO	The spindle–position coder gear ratio was incorrect.
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.
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
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. (Normally, this message is output when the setting of parameter No. 1828 is exceeded.)
SV1026	ILLEGAL AXIS ARRANGE	The parameter for servo axis arrange is not set correctly. A negative value, duplicate value, or greater value than the number of control axes was set to the parameter No. 1023 "The servo axis number of each axis."

Caution

⚠ CAUTION

1 While the operations for spindle positioning (canceling the spindle rotation mode to enter the spindle positioning mode, positioning the spindle in the spindle positioning mode, and canceling the spindle positioning mode to enter the spindle rotation mode) are being performed, automatic operation stop signal *SP is invalid. This means that even when the * SP signal is set to 0, automatic operation does not stop until the entire sequence is completed.
2 Dry run and machine lock are not enabled during spindle positioning.
3 For M codes used for the spindle positioning function, auxiliary function lock is not enabled.
4 Both serial spindle Cs contour control function (bit 2 (SCS) of parameter No.8133) and spindle positioning function (bit 1 (AXC) of parameter No. 8133) cannot be made effective at the same time.
If both are specified as AXC=1 and SCS=1, both functions become invalid.
Therefore, when a negative value is set in parameter No.1023 with above specification, alarm (SV1026) is generated.
Be sure to set AXC=1 and SCS=0 to use the spindle positioning function.
Be sure to set AXC=0 and SCS=1 to use the serial spindle Cs contour control function.
5 The spindle positioning axis is treated as one of the controlled axes. Therefore, signals related to controlled axes (such as an overtravel signal) must be set.
6 When analog spindles are used, set exact drift compensation values.
If an inappropriate drift compensation value is set, the positional deviation amount in the stopped state does not become 0. As a result, the following problems may occur when spindle orientation or spindle positioning is performed:
- In-position wait state is entered, which results in a stop.
- The spindle position is deviated. (The spindle does not reach or passes a specified position.)

- Note on using the rigid tapping function together

⚠ CAUTION

When using the rigid tapping function together with the spindle positioning function, do not specify rigid tapping in the spindle positioning mode, and contrariwise, do not specify spindle positioning in the rigid tapping mode.

Note

NOTE

1 M code commands related to spindle positioning must each be specified in an independent block. A block for specifying such an M code must not contain any other command. (An M command related to spindle positioning for another spindle must not be contained in the same block, either.)
Even when the function for enabling specification of multiple M codes in a single block is used together, spindle positioning M codes must be specified in an independent block.

2 The axis address for spindle positioning must be specified in an independent block. A block for specifying such an address must not contain any other command. The following commands, however, can be specified together with the axis address in the same block:
G00,G90,G91,G92 (M series, G code system B or C for T series)
G00,G50 (G code system A for T series)

3 M code commands related to spindle positioning are not buffered.

4 Spindle positioning cannot be performed by manual operations (such as a jog feed, manual handle feed, and manual numeric command).

5 Spindle positioning cannot be performed through PMC axis control.

6 Program restart/block restart cannot be performed for spindle positioning. Use MDI for specification.

7 The stored stroke limit check is disabled for the spindle positioning axis.

8 The pitch error compensation function is disabled for the spindle positioning axis.

9 When a setting is made to omit spindle orientation, the reference position return completion signal is not set to 1.

10 In spindle orientation, all-axes interlock/each-axis interlock checks are made only at the start of the block. Even when the signal is input in the middle of block execution, it is ignored.

11 The difference between a move command and the actual movement amount is maintained until the spindle positioning mode is canceled.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (B-64303EN-1)	Auxiliary function/2nd auxiliary function
	Spindle serial output/analog output
	Spindle speed control

FANUC SERVO MOTOR α*i* series MAINTENANCE MANUAL (B-65285EN)
FANUC SERVO AMPLIFIER α*i* series DESCRIPTIONS (B-65282EN)
FANUC AC SPINDLE MOTOR α*i* series PARAMETER MANUAL (B-65280EN)

10.11 Cs CONTOUR CONTROL

10.11.1 Cs Contour Control

Overview

The Cs contour control function positions the serial spindle using the spindle motor in conjunction with a dedicated detector mounted on the spindle.
This function can perform more accurate positioning than the spindle positioning function, and has an interpolation capability with other servo axes.

NOTE
1 Be sure to set 0 in bit 1 (AXC) of parameter No.8133 and 1 in bit 2 (SCS) of parameter No.8133 to use the serial spindle Cs contour control function.
2 Both serial spindle Cs contour control function and spindle positioning function cannot be made effective at the same time.
If both are specified as AXC=1 and SCS=1, both functions become invalid. Therefore, when a negative value is set in parameter No.1023 with above specification, alarm (SV1026) is generated.
3 Be sure to set 1 in bit 1 (AXC) of parameter No.8133 and 0 in bit 2 (SCS) of parameter No.8133 to use the spindle positioning function.
4 To use a serial spindle, set bit 5 (SSN) of parameter No. 8133 to 0. For details, see "SPINDLE SERIAL OUTPUT" mentioned previously.

- Increment system

As the least input increment of the Cs contour control axis, IS-B or IS-C must be set.
Selecting IS-C requires the following parameter settings:
Bit 1 (ISC) of parameter No.1013 = "1"
Bit 7 (CSC) of parameter No.3729 = "1"
Bit 0 (CS360M) of parameter No.4005 = "1"

- Command multiplication (parameter No.1820)

Command multiplication for the Cs contour control axis must be set to 2 (1×).

Explanation

The speed of the serial spindle is controlled by the spindle speed control function, while the spindle positioning is controlled by the Cs contouring control function ("spindle contour control"). Spindle speed control rotates the spindle using the velocity command, while the spindle contour control rotates the spindle using the move command.
Switching between spindle speed control and spindle contour control is performed by the DI signal from the PMC.
In the Cs contour control mode, the Cs contour control axis can be operated either manually or automatically, in the same way as normal servo axes.
(For a reference position return, see the relevant description in this section.)

- Setting the Cs contour control axis

Cs contour control is enabled for the first spindle in the path.
The axis used for Cs contour control must be set as an axis of the axes controlled by the CNC. Any controlled axis may be set as the Cs contour control axis.

Using parameter No. 1023, assign "-1" in the field corresponding to the chosen servo axis. (For single-path systems)

As the servo axis number (parameter No. 1023) of the axis used as the Cs contour control axis, set the number of the target logical spindle for Cs contour control with a minus sign (-1 to -n). (For 2-path systems)
This setting must be made only once for each control system. The spindle operating as the Cs contour control axis is the first spindle in the path.

The axis used as the Cs contour control axis must be set as a rotation axis (bit 0 (ROTx) of parameter No. 1006 and parameter No. 1022)

- Command Address

The address for the move command in Cs contour control is the axis name specified in parameter No.1020. This address is arbitrary.
When the second auxiliary function is provided (bit 2 (BCD) of parameter No. 8132 is set to 1), the address used for specifying the second auxiliary function (B for the T series; for the M series, an address specified in parameter No. 3460) cannot be used as the axis name. For the T series, when address A or C is to be used as the axis name, set bit 4 (CCR) of parameter No. 3405 to 0.

- Switching between spindle speed control and Cs contour control

- Switching from spindle speed control to Cs contour control
  When the Cs contour control change signal CON <G027.7> is 1, the first spindle is regarded as the Cs contour control axis.

When the mode is switched while the spindle is rotating, the spindle stops immediately to perform the change.

- Switching from Cs contour control mode to spindle speed control mode
  When the Cs contour control change signal CON <G027.7> is 0, each spindle is placed in the spindle speed control mode.

Confirm that the move command for the spindle has been completed, then specify the switch. If it is switched while the spindle is moving, the machine enters interlock, or excessive position deviation alarm occurs.

- Setting axes for interpolation with a Cs contour control axis

Up to four servo axes can be used for interpolation with a Cs contour control axis. Only linear interpolation is possible. Any servo axes can be selected for interpolation with the Cs contour control axis by parameter setting. Set parameters as follows:

- When no servo axis is used for interpolation, specify "0" in parameter Nos. 3900, 3910, 3920, and 3930.
- When one or more servo axes are used for interpolation, set the parameter for each as follows :

(1) Assign the axis number (1 to 5) to each of the servo axes used for interpolation in parameter nos. 39n0 (n=0, 1, 2, or 3).
(2) Set the loop gain for each of the servo axes specified in (1) in parameter nos. 39n1, 39n2, 39n3. The loop gain must be the position loop gain for the Cs contour control axis or a desired value. Four parameters are provided to correspond to the four gears of the spindle. Use those parameters according to the inputs of the serial spindle clutch /gear signal CTH1A, CTH2A <G70.3, 2>.
(3) When the number of servo axes to be used for interpolation is smaller than four, set "0" in remaining parameter Nos. 39n0.

When bit 7 (ALG) of parameter No. 1814 is set to 1, the loop gain can be set automatically at the time of switching between spindle speed control and Cs contour control.

- Automatic setting of the loop gain at the time of switching between spindle speed control and Cs contour control
  1 Switching from spindle speed control to Cs contour control
    The position loop gain of the Cs contour control axis selected by the clutch/gear signals (CTH1 and CTH2) (one of parameter Nos. 4069, 4070, 4071, and 4072) is set automatically for the servo axis for which bit 7 (ALG) of parameter No. 1814 is set to 1.
  2 Switching from Cs contour control to spindle speed control
    The original position gain (parameter No. 1825) is set automatically for the servo axis.
  3 Switching between spindle speed control and Cs contour control during automatic operation
    Switching is impossible during automatic operation and during axis movement.
    If switching from spindle speed control to Cs contour control or vice versa is made in the middle of a block executed in automatic operation, position gains do not change immediately; they are set automatically after all controlled axes are stopped completely.
  4 Similarly, when switching between spindle speed control and Cs contour control is made in other than automatic operation, automatic setting is performed after the completely stopped states of all controlled axes are confirmed

- Reference Position Return of Cs Contour Control Axis

After the serial spindle is switched from spindle speed control to Cs contour control mode, the current position is undefined. Return the spindle to the reference position.
The reference position return of the Cs contour control axis is as follows:

- In manual mode
  After the serial spindle enters the Cs contour control mode, move the spindle in the direction of the reference position by turning on the feed axis and direction select signal (+Jn (G100) or -Jn (G102)). The spindle starts the movement to the reference position; when it reaches that position, the reference position return completion signal ZPn <F094> turns to "1".
  Turning any feed axis and direction select signal to "1" moves the tool in the reference position direction.
- In the automatic mode
  (i) G00 command
    If a serial spindle is placed in the Cs contour control mode, then the G00 command is issued without performing reference position return even once when bit 1 (NRF) of parameter No. 3700 is 0, the serial spindle performs a reference position return.
    As G00 is specified, the serial spindle indexes the reference position, then performs positioning at a specified position.
    Only when positioning at the reference position is performed, the reference position return completion signal ZPn <F094> is set to 1 after completion of the positioning operation.
    When a reference position return has been performed at least once, a G00 command performs a normal positioning operation.
    When a reference position return is performed manually or by using G28, the spindle is always positioned at the reference position; on the other hand, when a reference position return is performed by using G00, the spindle can be positioned at any specified position.
  (ii) G28 command
    After the serial spindle is put in the Cs contour control mode, issuing the G28 command stops the spindle motor, then moves the spindle to the midpoint. The spindle then returns to the reference position. At this point, the reference position return completion signal ZPn <F094> turns to "1". When the serial spindle has returned to the reference position once while in the Cs contour control mode, the G28 command positions the spindle at the reference position without moving to the midpoint and ZPn comes on.
- Interruption of reference position return
  (i) Manual operation
    Return to the reference position can be interrupted by resetting, emergency stop, or turning off the feed axis and direction select signal. When the interrupted return operation is resumed, start from the beginning.

(ii) Automatic operation
Return to the reference position can be interrupted by resetting, emergency stop, or feed hold. When the interrupted return operation is resumed, start from the beginning.

- Operation of Cs contour control axis (Manual/Automatic)

If a reference position return is performed on the Cs contour control axis, the axis can be operated in the same way as a normal NC axis.

In the spindle speed control mode, on the other hand, it does not operate as the Cs contour control axis, and alarm PS0197 occurs during automatic operation.
In the spindle speed mode, inhibit manual operation of the Cs contour control axis using the PMC ladder.

Diagnosis display

- Display of Position Error of Cs Contour Control Axis

418	**Position deviation amount of each spindle**

Position deviation amount of the position loop for the each spindle.
This diagnostic display shows information obtained from the serial spindle control unit. This diagnosis displays position error of the spindle contour axis during spindle contour control.
The position error can also be checked using a servo error display (DGN of No. 300) for an axis under Cs contour control.

300	**Position deviation amount of each servo axis**

Signal

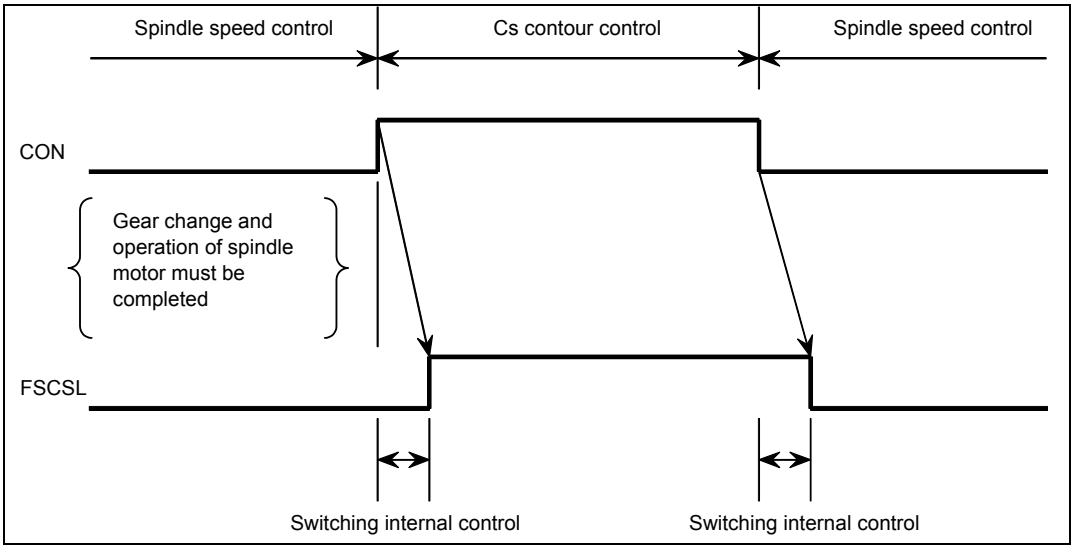
Cs contour control change signal CON<Gn027.7>

[Classification] Input signal
[Function] This signal specifies that the first spindle be switched between the spindle speed control and Cs contour control modes.

Cs contour control change completion signal FSCSL<Fn044.1>

[Classification] Output signal
[Function] This signal posts that the first spindle has switched to the Cs contour control mode.
[Output cond.] This signal is set to 1 when:
The spindle switches to the Cs contour control mode.
The signal is set to 0 when:
The spindle is not in the Cs contour control mode (is in the spindle speed control mode).

- **Time Chart**



NOTE
1 Any mechanical gear change needed and inputs for GR1, GR2, CTH1A, and CTH2A must be completed before the CON signal selects Cs contour control mode.
2 A servo excessive error may be generated if the spindle motor is not ready for operation. (Signal SRVA, SFRA <G070.4, 5> or other required signals must be appropriately processed on the machine side).

- **Other signals**

Gear select signals GR1O,GR2O,GR3O<Fn034.0 to 2>

[Classification] Output signal
[Function] This signal is output to the PMC to specify the gear stage.
[Output cond.] For information about this signal, see "SPINDLE SPEED CONTROL".

Gear select signals GR1,GR2<Gn028.1,2>

[Classification] Input signal
[Function] This signal informs the CNC of the gear stage currently selected.
[Operation] For information about this signal, see "SPINDLE SPEED CONTROL".

Clutch/Gear signal (Serial spindle)

CTH1A,CTH2A<Gn070.3,Gn070.2> : First spindle

Refer to the manual of serial spindle.

These signals determine what parameter (loop gain, etc.) to be used for each gear position.
CTH1 and CTH2 are the gear select signals for the serial spindle, but GR1 and GR2 must also be set. Do not change these signals while in the Cs contour control mode.

Relationship between gears selected and spindle gear select signals

CNC side							Serial spindle	
T/M series with CSSC			M series without CSSC					
GR2	GR1	Gear selection	GR3O	GR2O	GR1O	Gear selection	CTH1	CTH2
0	0	1st stage	0	0	0	1st stage	0	0
0	1	2nd stage	0	1	1	2nd stage	0	1

CNC side							Serial spindle	
T/M series with CSSC			M series without CSSC					
1	0	3rd stage	1	0	0	3rd stage	1	0
1	1	4th stage					1	1

CSSC: Constant surface speed control

NOTE
1 When the M series does not include the constant surface speed control, and parameter No. 3706#4 GTT=0, GR1 and GR2 do not need to be input. Input CTH1 and CTH2 when gears are changed using GR1O, GR2O and GR3O.
2 The above combination of clutch/gear signals CTH1 and CTH2 is an example.
The serial spindle gear is selected by CTH1 and CTH2 independently of gear selection on the CNC side. So, enter necessary signals, and set the corresponding serial spindle parameters.

Cs contour control axis reference position return completion signals ZP1 to ZP5 <Fn094.0 to Fn094.4>

[Classification] Output signal

[Function] This signal indicates that a reference position return has been made for the Cs contour control axis.

ZPx

x : 1 ..... Reference position return completion signal of 1st spindle
2 ..... Reference position return completion signal of 2nd spindle
3 ..... Reference position return completion signal of 3rd spindle
: :

[Output cond.] If a manual reference position return or automatic reference position return by G28 is performed during the Cs contour control mode, this signal becomes logical 1 when the Cs contour control axis reaches the reference position.

Signals on manual operation

Feed axis and direction select signal +Jn, -Jn <G100, G102> (Input) Manual handle feed axis select signal HSnA, HSnB, HSnC, HSnD <G018, G019> (Input) (Refer to respective items in this manual)
The Cs contour control axis can be manually operated in the same way as normal servo axes, except for a manual reference position return. In the spindle speed control mode, however, manual operations for the Cs contour control axis must be inhibited using the PMC ladder, etc.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn027	CON							
Gn028						GR2	GR1	
Fn034						GR3O	GR2O	GR1O
Fn044							FSCSL	
Fn094				ZP5	ZP4	ZP3	ZP2	ZP1

- For first serial spindle

	#7	#6	#5	#4	#3	#2	#1	#0
Gn070	MRDYA		SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
Fn045			LDT2A	LDT1A				

Parameter

Major related parameters are described below.
In addition to the parameters described below, axis speed, acceleration, indication, and other parameters can be used. It is not necessary to set the digital servo parameters (parameter Nos. 2000 and up) for the axis used as the Cs contour control axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1006					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 rotation axis.
#1 ROSx

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	Rotation 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 parameter No.1008#0(ROAx) and #2(RRLx). (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	Rotation axis (B type) (1) Inch/metric conversion, absolute coordinate values and relative coordinate values are 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 rotation axis roll over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

Set a Cs contouring control axis as a rotation axis.

NOTE
1 Inch/metric conversion is not performed on rotation axes.
2 For rotation axes, the machine coordinate system is normalized to the range from 0 to 360 degrees. Automatic reference position return (G28, G30) is done in the direction of manual reference position return and the amount of travel does not exceed one turn.

#3 DIAx The move command for each axis is based on:
0: Radius specification
1: Diameter specification

For a Cs contouring control, set 0.

NOTE
For the FS0*i*-C, one of the following changes is required besides setting bit 3 (DIAx) of parameter No. 1006 so that the axis based on diameter specification achieves the specified amount of movement.
- Halve the command multiplication (the detection unit is not changed).
- Halve the detection unit and double the flexible feed gear (DMR).

For the FS0*i*-D, only if bit 3 (DIAx) of parameter No. 1006 is set, the CNC halves the specified pulse. Accordingly, the above changes are not required (when the detection unit is not changed). To halve the detection unit, double both CMR and DMR.

	#7	#6	#5	#4	#3	#2	#1	#0
1013							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 ISA
#1 ISC Increment system of each axis

Increment system	#1 ISC	#0 ISA
IS-A	0	1
IS-B	0	0
IS-C	1	0

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), 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	Rotation 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

Setting	Meaning
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.

For a Cs contouring control, set 0.

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 Number of controlled axes
Set the servo axis for each control axis.
Usually set to same number as the control axis number.
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 control axis by using the first spindle, set -1.

1260 **Amount of a shift per one rotation of a rotation 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 amount of a shift per one rotation of a rotation axis.
For the rotation axis used for cylindrical interpolation, set the standard value.

For a Cs contouring control, set 360.0.

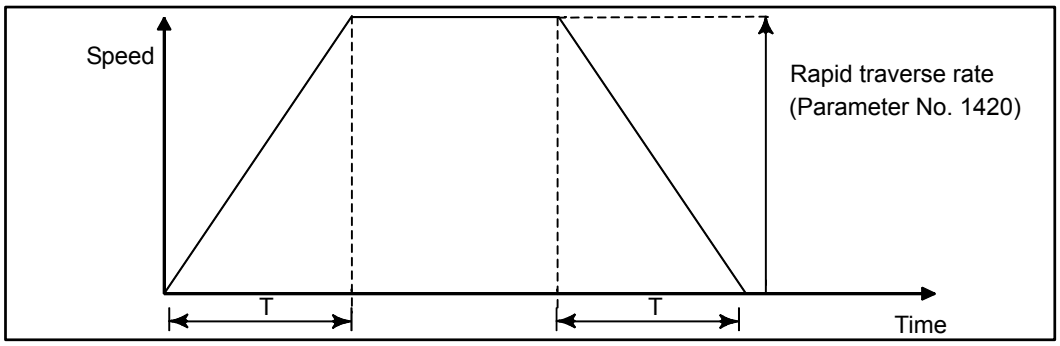
1420 **Rapid traverse rate for each axis**

[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.

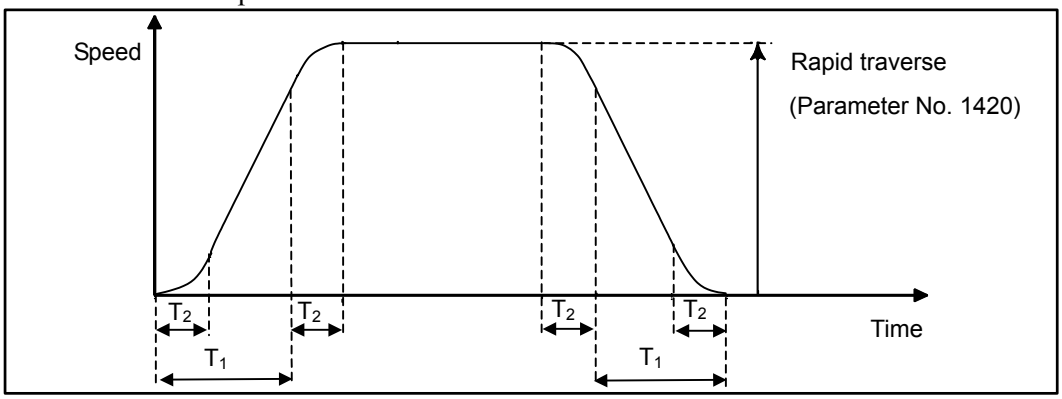
1620	Time constant T or T1 used for linear acceleration/deceleration or bell-shaped acceleration/deceleration in rapid traverse for each axis

[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 : $T_2 \times 2$

1621	Time constant T2 used for 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 1000
Specify time constant T2 used for bell-shaped acceleration/ deceleration in rapid traverse for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1814	ALGx							

[Input type] Parameter input
[Data type] Bit axis

#7 ALGx The servo axis loop gain in the Cs contour control mode is:
0: Not matched with the Cs contour control loop gain.
1: Matched with the Cs contour control loop gain.

NOTE
1 For those axes that are used for electronic gear box (M series), spindle positioning (T series), index table indexing (M series), PMC axis control, rigid tapping, and so forth, set this parameter to 0 (Not matched with the Cs contouring control loop gain).
2 When specifying axes subject to interpolation with the Cs contouring control axis by using parameter No. 39n0 (n = 0 to 3), set this parameter to 0.
3 When making a setting so that a position gain is automatically set at the time of Cs contouring control switching by setting bit 7 (ALG) of parameter No. 1814 to 1, set all of parameter No. 39n0 (n = 0 to 3) 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
[Unit of data] 0.5 times
[Valid data range] See below :
For a Cs contouring control, set 2.

1825	Servo loop gain for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 0.01/sec
[Valid data range] 1 to 9999
Set the loop gain for position control for each axis.

NOTE
If there is a servo axis that is subject to interpolation with the Cs contouring control axis, set parameters No. 3900 to No. 3944 to match the loop gain of the Cs contouring control axis to that of the servo axis in the Cs contour control mode.

1826	In-position width for each axis

[Input type] Parameter input

[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
The in-position width is set for each axis.
When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in-position width, the machine is assumed to have reached the specified position. (The machine is in the in-position state.)

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) 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) is generated, and operation is stopped immediately (as in emergency stop).

1830	Axis-by-axis positional deviation limit at servo-off time

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
This parameter is used to set a positional deviation limit at servo-off time, on an axis-by-axis basis.
If the value specified with this parameter is exceeded at servo-off time, a servo alarm is issued to cause an immediate stop (same as an emergency stop). Usually, set the same value as a positional deviation at stop time.

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.

	#7	#6	#5	#4	#3	#2	#1	#0
3700							NRF	

[Input type] Parameter input
[Data type] Bit path

#1 NRF With the first move command (G00) after switching the serial 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.

NOTE
1 When using the Cs axis establishment function, this parameter is recommended to be set to 1.
2 The setting of this parameter is valid for G00. The first rapid traverse of a canned cycle is normal positioning regardless of the setting of this parameter.

	#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.

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.
3: Spindle motor connected to amplifier number 3 is used.

NOTE
When using an analog spindle, set it at the end of the spindle configuration.
(Example)
When there are three spindles in an entire system (two serial spindles and one analog spindle), set the spindle amplifier number (this parameter) of the analog spindle to 3.

	#7	#6	#5	#4	#3	#2	#1	#0
3729	CSCs				NCSs	CSNs		

[Input type] Parameter input
[Data type] Bit spindle

#2 CSNs When the Cs contour control mode is turned off, an in-position check is:
0: Performed.
1: Not performed.

NOTE
If this parameter is set to 1, the same operation as in FS0*i*-C is assumed.

#3 NCSs When the Cs contour control mode is set:
0: Switching to Cs contour control is completed when the spindle activating current is on (the spindle amplifier is ready for operation in the Cs contour control mode).
1: Switching to Cs contour control is completed even when the spindle activating current is off (the spindle amplifier is not ready for operation in the Cs contour control mode).

If this parameter is set to 1, the Cs contour control switch end signal is output without waiting for the spindle to decelerate to a stop.

#7 CSC The increment system of the Cs contour control axis is:
0: IS-B.
1: IS-C.

3900	Number of servo axis for interpolation with Cs contour control axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to number of controlled axes
Set the number of the servo axis to be used for interpolation with the Cs contour control axis. (For group 1)

NOTE
When there is no servo axis used for interpolation with the Cs contour control axis, set 0.

3901	Loop gain for servo axis for interpolation with Cs contour control axis (HIGH gear)

3902	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM HIGH gear)

3903	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM LOW gear)

3904	Loop gain for servo axis for interpolation with Cs contour control axis (LOW gear)

[Input type] Parameter input
[Data type] Word path
[Unit of data] 0.01/sec
[Valid data range] 0 to 9999
Set the servo loop gain for each spindle gear of the servo axis used for interpolation with the Cs contour control axis. (For group 1)

3910	Number of servo axis for interpolation with Cs contour control axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to number of controlled axes
Set the number of the servo axis to be used for interpolation with the Cs contour control axis. (For group 2)

NOTE
When there is no servo axis for interpolation with the Cs contour control axis, or when there is not more than one servo axis, set 0.

3911	Loop gain for servo axis for interpolation with Cs contour control axis (HIGH gear)

3912	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM HIGH gear)

3913	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM LOW gear)

3914	Loop gain for servo axis for interpolation with Cs contour control axis (LOW gear)

[Input type] Parameter input
[Data type] Word path
[Unit of data] 0.01/sec
[Valid data range] 0 to 9999
Set the servo loop gain for each spindle gear of the servo axis used for interpolation with the Cs contour control axis. (For group 2)

3920	Number of servo axis for interpolation with Cs contour control axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to number of controlled axes
Set the number of the servo axis to be used for interpolation with the Cs contour control axis. (For group 3)

NOTE
When there is no servo axis for interpolation with the Cs contour control axis, or when there is not more than two servo axes, set 0.

3921	Loop gain for servo axis for interpolation with Cs contour control axis (HIGH gear)

3922	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM HIGH gear)

3923	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM LOW gear)

3924	Loop gain for servo axis for interpolation with Cs contour control axis (LOW gear)

[Input type] Parameter input
[Data type] Word path
[Unit of data] 0.01/sec
[Valid data range] 0 to 9999
Set the servo loop gain for each spindle gear of the servo axis used for interpolation with the Cs contour control axis. (For group 3)

3930	Number of servo axis for interpolation with Cs contour control axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to number of controlled axes
Set the number of the servo axis to be used for interpolation with the Cs contour control axis. (For group 4)

NOTE
When there is no servo axis for interpolation with the Cs contour control axis, or when there is not more than three servo axes, set 0.

3931	Loop gain for servo axis for interpolation with Cs contour control axis (HIGH gear)

3932	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM HIGH gear)

3933	Loop gain for servo axis for interpolation with Cs contour control axis (MEDIUM LOW gear)

3934	Loop gain for servo axis for interpolation with Cs contour control axis (LOW gear)

[Input type] Parameter input
[Data type] Word path
[Unit of data] 0.01/sec
[Valid data range] 0 to 9999
Set the servo loop gain for each spindle gear of the servo axis used for interpolation with the Cs contour control axis. (For group 4)

	#7	#6	#5	#4	#3	#2	#1	#0
4000					CSO			

[Input type] Parameter input
[Data type] Bit spindle
#3 CSO Direction of reference position return in Cs contour control
0: Reference position return is performed by rotating the spindle counterclockwise (when viewed from the motor output axis).
1: Reference position return is performed by rotating the spindle clockwise (when viewed from the motor output axis).

4021	Maximum speed in Cs contour control

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] min^{-1}
Set the maximum speed of the spindle in Cs contour control.

4056	Gear ratio (HIGH gear)

4057	Gear ratio (MEDIUM HIGH gear)

4058	Gear ratio (MEDIUM LOW gear)

4059	Gear ratio (LOW gear)

[Input type] Parameter input
[Data type] Integral word spindle
[Unit of data] Motor speed per spindle rotation × 100
[Valid data range] 0 to 32767
Set the gear ratio of the spindle to spindle motor.

NOTE
When a serial spindle is positioned, set the gear ratio of the spindle to spindle motor.
The gear ratio to be used for spindle operations is set by clutch/gear signals (serial spindle) CTH1s and CTH2s.

4069	Position gain in Cs contour control (HIGH gear)

4070	Position gain in Cs contour control (MEDIUM HIGH gear)

4071	Position gain in Cs contour control (MEDIUM LOW gear)

4072	Position gain in Cs contour control (LOW gear)

[Input type] Parameter input
[Data type] Integral word spindle
[Unit of data] 0.01/sec
[Valid data range] 0 to 32767
Set the position gain in Cs contour control.

4074	Reference position return speed in servo mode

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] min^{-1}
As the spindle orientation speed in a servo mode (spindle positioning, rigid tapping, and so on), the spindle speed set in this parameter is used.

NOTE
1 When a serial spindle is positioned, the orientation (reference position return) speed depends on this parameter.
2 If this parameter is set to 0, the reference position return speed is assumed to be the setting of parameter No. 4021.

4135	Grid shift amount in Cs contour control

[Input type] Parameter input
[Data type] Integral 2-word spindle

[Unit of data] Detection unit
[Valid data range] Within ±1 rotation (For 1-micron detection, for example, -360000 to 360000)
Set the grid shift amount (the distance from the one-rotation signal position to machine zero point) in Cs contour control.

	#7	#6	#5	#4	#3	#2	#1	#0
8133			SSN			SCS		

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#2 SCS Cs contour control is:
0: Not Used.
1: Used.

#5 SSN Spindle serial output is:
0: Used.
1: Not Used.

Alarm and message

Number	Message	Description
PS0194	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 simple spindle synchronous control mode.
PS0197	C-AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cs-axis when the Cs contour control switching signal was off.
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.

Caution

⚠ CAUTION
1 The M codes to specify Cs contour control mode switching must be assigned to M codes not buffered. (Parameter Nos. 3411 to 3420)
2 Avoid changing spindle gears in Cs contour control mode. When a gear change is required, be sure to perform the gear change in spindle speed control mode.
3 The following error compensation functions are enabled for the Cs contour control axis:
Pitch error compensation, both-direction pitch error compensation, backlash compensation

⚠ CAUTION

4 For the axis under Cs contour control, the following functions cannot be used:
- Index table indexing (M series)
- Polar coordinate command (M series)
- Axis synchronous control
- Axis control by PMC
- Removal of controlled axis

5 In the Cs contour control mode, when an axis control parameter for the Cs contour control axis such as the acceleration/deceleration time constant is changed, the new parameter value does not become valid immediately. The new parameter value becomes valid after the Cs contour control mode is canceled then entered again.

6 When the position gain is automatically set at Cs contour control switching (when bit 7 (ALG) of parameter No. 1814 is set to 1), changing the setting in parameter No. 4069 to 4072 of the loop gain corresponding to the selected spindle gear does not make the new setting valid immediately.
To make the new setting valid, cancel the Cs contour control mode, then set the Cs contour control mode again. If more than one Cs contour control axis is present, however, the newly set value is included in the position gain values selected for these axes among which the smallest value is to be selected.

Reference item

For details of the parameters (parameter Nos. 4000 to 4539), signals, and alarms for the serial spindle control unit, refer to the following manuals of serial spindles:
FANUC AC SPINDLE MOTOR α*i* series PARAMETER MANUAL (B-65280EN)
FANUC SERVO AMPLIFIRER α*i* series DESCRIPTIONS (B-65282EN)
FANUC SERVO SYSTEM α*i* series MAINTENANCE MANUAL (B-65285EN)

10.11.2 Cs Contour Control Torque Limit Skip

Overview

This function enables a torque limit skip for the Cs contour control axis.

Explanation

Torque limit command signal TLMHA<Gn070.1> and load detection signal LDT1A<Fn045.4> for serial spindles are used to make a torque limit skip for the Cs contour control axis.
With this function, if load detection signal LDT1A<Fn045.4> of a serial spindle operated under Cs contour control is set to 1 when a torque limit skip is specified (G31 P98/P99), skip processing is performed.

To use this function, make the following settings:
(1) Set bit 0 (CST) of parameter No. 6215 for the corresponding Cs contour control axis to 1.
(2) Set a torque limit in parameter No. 4025. (Unit: %, maximum torque ratio)
(3) Set torque limit command signal TLMHA<Gn070.1> to 1. In this signal state, the output torque is limited to the value set in parameter No. 4025.
(4) Set the load detection level in parameter No. 4026. This setting must be slightly smaller than the torque limit set in step (2) above.

When this load detection level is reached, load detection signal LDT1A<Fn045.4> is output, enabling skip processing if a torque limit skip is specified (G31 P98/P99).

The other specifications are the same as for normal torque limit skip. For details of the torque limit skip, refer to the description of torque limit skip in "Connection Manual (Function) (this manual)" and the description of torque limit skip in "Operator's Manual".
For torque limit command signal TLMHA<Gn070.1> and load detection signal LDT1A<Fn045.4> for serial spindles, refer to relevant manuals such as "FANUC Spindle Motor αi series PARAMETER MANUAL" (B-65280EN).

NOTE
PMC sequence programs must be created so that Cs contour control axes are always placed in the torque limit state before the G31 P99/P98 is specified.

```
(Sample program)
O0012
:
Mxx (Specify torque limit from PMC)
:
G31 P99 C200.F100. (Torque limit skip command)
:
Mxx (Cancel torque limit from PMC)
:
M30
```

Signal

Torque limit command signal (serial spindle)
TLMHA<Gn070.1> : First spindle

[Classification] Input signal
[Function] These signals limit the output torque of the corresponding serial spindle to the value set in parameter No. 4025. To cause a torque limit skip for a Cs contour control axis, set this signal to 1. For details, refer to the relevant manual of the serial spindle.

Load detection signal (serial spindle)
LDT1A <Fn045.4> : First spindle

[Classification] Output signal
[Function] These signals are output when the load on the corresponding serial spindle has reached the value set in parameter No. 4026.
For details, refer to the relevant manual of the serial spindle.

Torque limit state signal (serial spindle)
TLMA <Fn045.6> : First spindle

[Classification] Output signal
[Function] These signals are output when the torque is being limited by the corresponding torque limit command signal.
For details, refer to the relevant manual on the serial spindle.

Signal address
- **For first serial spindle**

	#7	#6	#5	#4	#3	#2	#1	#0
Gn070							TLMHA	
Fn045		TLMA		LDT1A				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6215								CSTx

[Input type] Parameter input
[Data type] Bit axis

#0 CSTx On a Cs contour control axis, torque limit skip operation is:
0: Not performed.
1: Performed.
Torque limit skip operation is performed using the torque limit command signal TLMH and the load detection signal LDT1 of the serial spindle.

NOTE
When setting this parameter to carry out a torque limit skip on a Cs contour control axis, keep the following in mind.
1 Set bit 4 of the serial spindle parameter of the Cs contour control axis (spindle) that uses the torque limit skip function to 1 so that the load detection signal is output during acceleration/deceleration.
2 If the load detection state is entered (LDT1 = "1") when the torque limit command is executed (TLMH1 = "1") in the Cs mode, the alarm detection level in the stop state is not checked on the axis.
3 If the load detection state is entered (LDT1 = "1") in the Cs mode, an in-position check is not performed on the axis.

4025	Torque limit value

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%
[Valid data range] 0to100
Set a torque limit value applicable when a torque limit command (TLMH) is specified. Specify limit value data, assuming that the maximum torque represents 100%.

4026	Load detection level 1

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 1%
[Valid data range] 0 to 100
Set the detection range of the load detection signal (LDT1).
When the motor output exceeds (setting data)% of the maximum output, the load detection signal LDT1 is set to 1.

NOTE
1 If torque limit skip is specified (with bit 0 of parameter No. 6215 set to 1) for a Cs contouring control axis, an excessive error check during a stop is not made for the axis when the load detection state (LDT1 = 1) is set while the torque limit command is specified (TLMH = 1) in the Cs contouring control mode.

NOTE
2 If torque limit skip is specified (with bit 0 of parameter No. 6215 set to 1) for a Cs contouring control axis, an in-position check is not made for the axis when the load detection state (LDT1 = 1) is set in the Cs contouring control mode.

	#7	#6	#5	#4	#3	#2	#1	#0
4009				LDTOUT				

[Input type] Parameter input
[Data type] Bit spindle

 # 4 LDTOUT During acceleration/deceleration, the load detection signal is:
0: Not output.
1: Output.

10.11.3 Cs Contour Control Axis Coordinate Establishment

Overview

Shifting a serial spindle from spindle rotation control to Cs contour control results in its current position being lost.
This function is intended to establish the current position without making a reference position return. This is done by setting the Cs axis coordinate establishment request signal CSFI1<Gn274.4> to '1'.

NOTE
Once the power has been turned on, this function remains enabled until the power is turned off after the return of the Cs contour axis to the reference position.

Explanation

- Setting

This function is enabled by setting bit 2 (CSF) of parameter No. 3712 and bit 5 (CSPTRE) of parameter No. 4353 to 1.
(Using this function requires resetting bit 7 (RFCHK3) of parameter No.4016 to 0.)

Inhibit buffering with the M code (parameter Nos. 3411 to 3432) used to set the Cs axis coordinate establishment request signal CSFI1<Gn274.4> to "1".

- Cs axis coordinate establishment procedure

(1) Turn the Cs contour control mode ON, using the M code as a trigger, and set the Cs axis coordinate establishment request signal CSFI1<Gn274.4> to "1".
(2) The absolute, relative, and machine coordinate systems for the Cs axis are established if the Cs axis origin established state signal CSPENA<Fn048.4> is "1".
(Establishing the relative coordinate system requires setting bit 3 (PPD) of parameter No. 3104 to 1.)
(3) Once the coordinate systems are established, the reference position established signal ZRFx<Fn0120.0 to 4> becomes "1". So, reset the Cs axis coordinate establishment request signal CSFI1<Gn274.4> to "0".
(4) Perform the FIN processing for the M code.

NOTE
1 When inputting the MRDY, SFR, and SRV signals from the PMC to serial spindles via the CNC, ensure they are input normally.

NOTE
2 In an emergency stop or servo alarm condition, no Cs axis coordinate system can be established.
3 Do not shift the Cs contour axis when Cs axis coordinate establishment is under way.

- Case in which no Cs axis coordinate establishment can be performed

If Cs axis coordinate establishment cannot be performed, alarm PS5346 is issued for the path where the absolute coordinates (workpiece coordinates) are is to be established.

The following conditions prevent Cs axis coordinate establishment.

- The Cs axis origin established state signal CSPENA<Fn048.4> is "0".
- An attempt to transfer positional information from the spindle amplifier has failed.
- The servo section is off when an attempt is made to start Cs axis coordinate establishment.
- The Cs axis is under synchronous control or superimposed control. (T series)
- An emergency stop condition has occurred during coordinate establishment.
- The Cs axis has exited composite control during coordinate establishment. (T series)
- An attempt has been made to start synchronous, composite, or superimposed control for the Cs axis during coordinate establishment. (T series)

The Cs axis coordinate establishment alarm signal CSFO1<Fn274.4> becomes "1" in the path to which the spindle is connected. So, reset the Cs axis coordinate establishment request signal CSFI1<Gn274.4> to "0".
Resetting the Cs axis coordinate establishment request signal CSFI1<Gn274.4> to "0" causes the Cs axis coordinate establishment alarm signal CSFO1<Fn274.4> to be reset to "0".

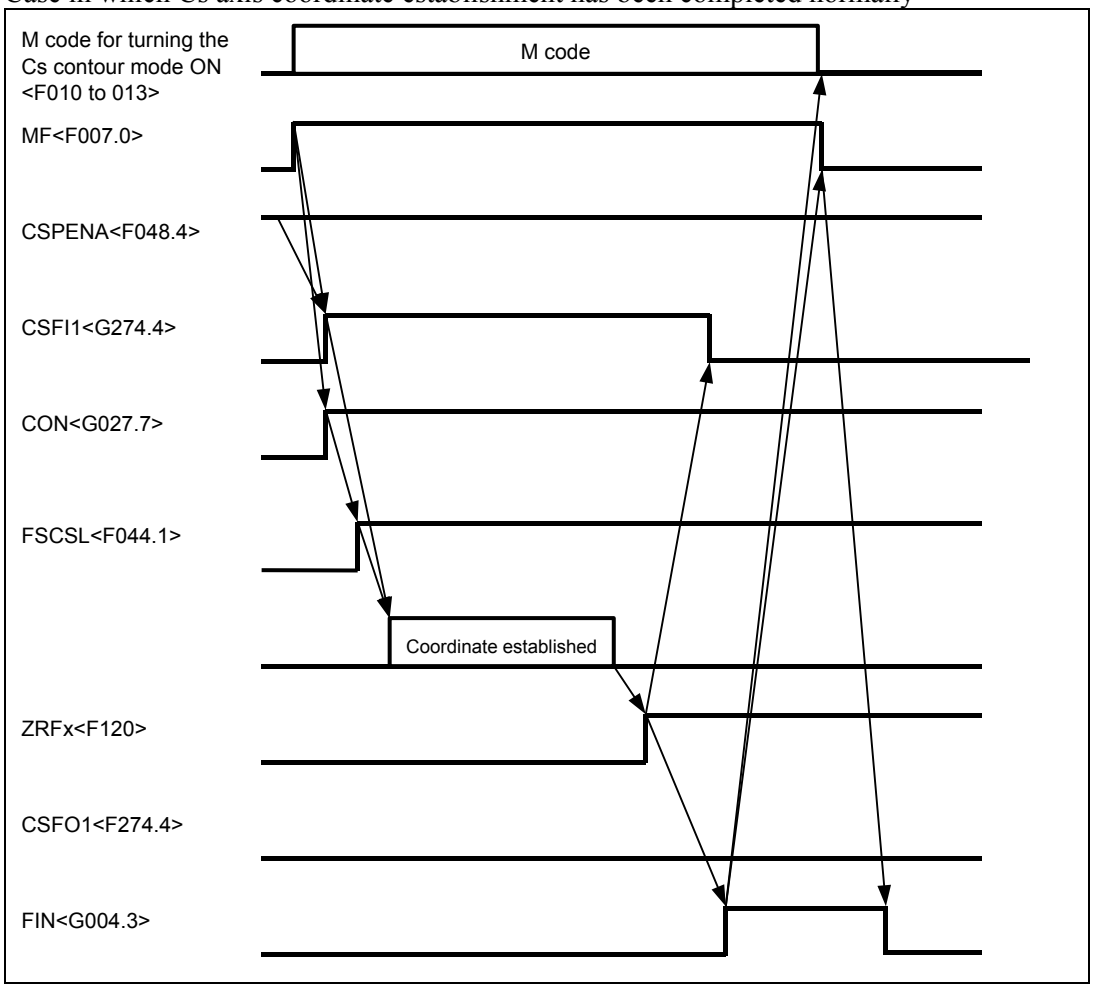
- Clearing PS alarm 5346

Alarm PS5346 cannot be cleared before the Cs axis reference return position is established.
In the path where alarm PS5346 occurred, make a manual reference position return for the Cs axis.
Once the reference position has been established, a reset can clear alarm PS5346.

Performing a reset after the Cs contour control mode is exited enables alarm PS5346 to be cleared.

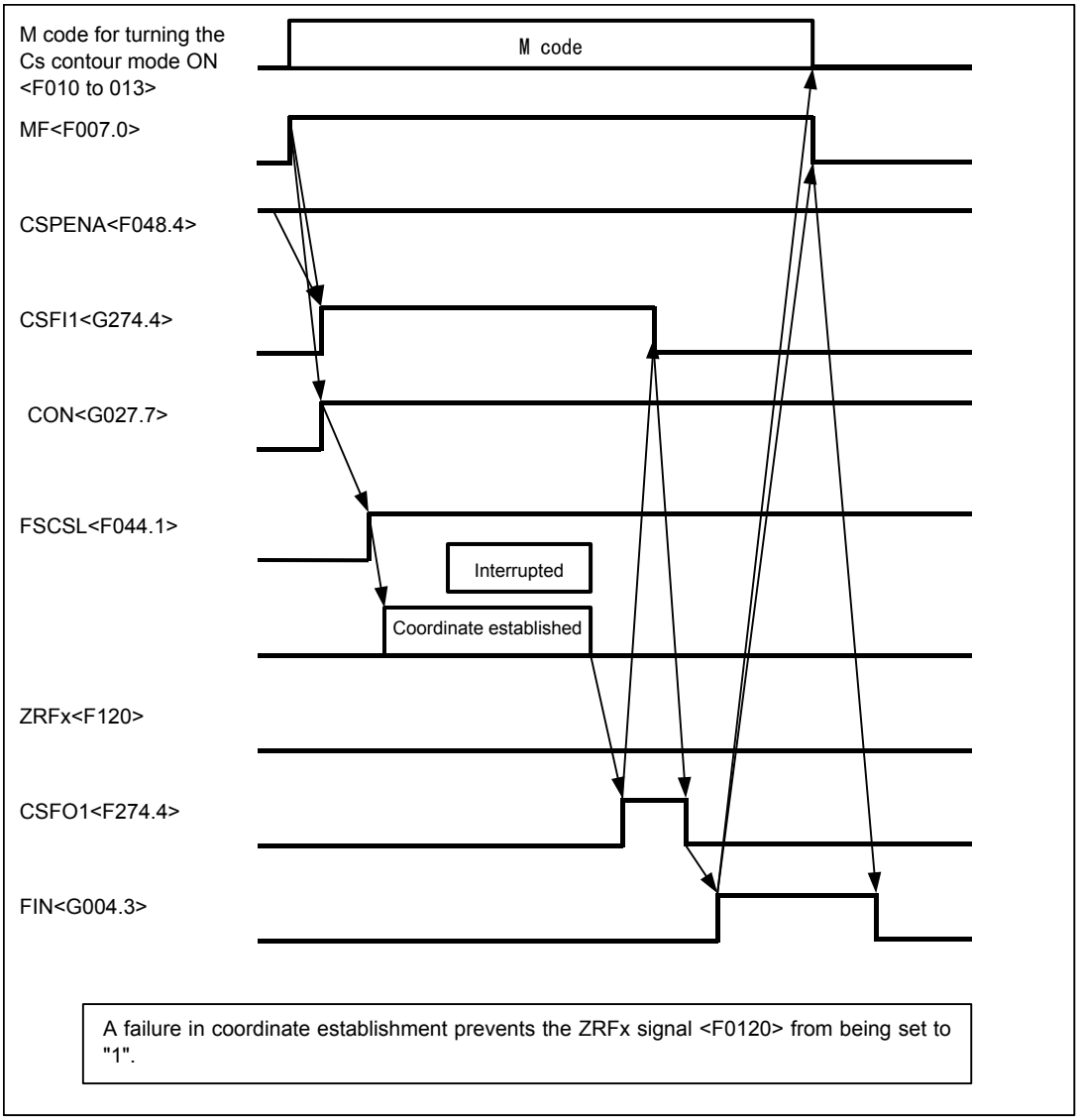
- Sequence (timing chart)

Case in which Cs axis coordinate establishment has been completed normally

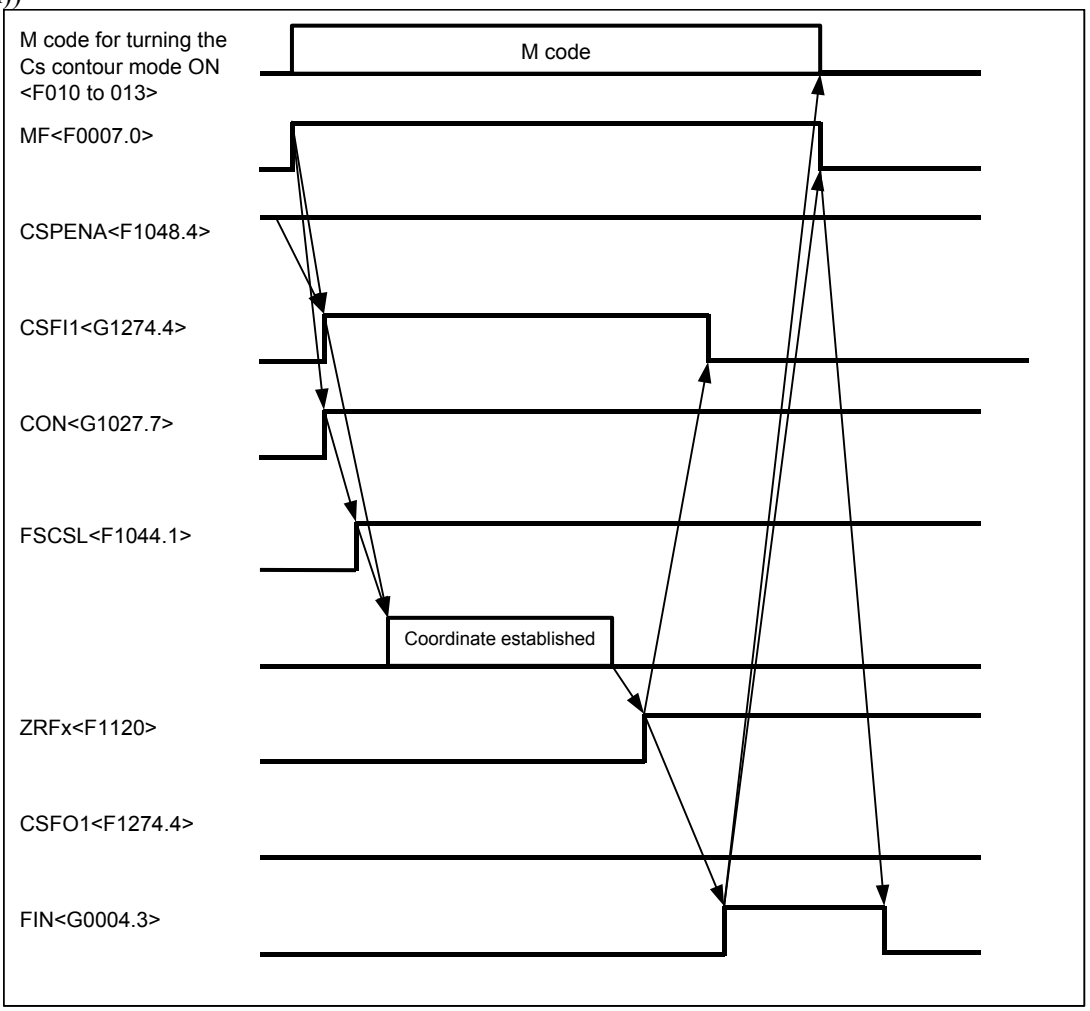


Cs axis coordinate establishment remains enabled in the Cs contour mode.
In any mode other than the Cs contour mode, setting the Cs axis coordinate establishment request signal CSFI1<Gn274.4> to "1" causes Cs axis coordinate establishment to be ignored.

Case in which Cs axis coordinate establishment cannot be achieved


M code for turning the Cs contour mode ON <F010 to 013>
M code
MF<F007.0>
CSPENA<F048.4>
CSFI1<G274.4>
CON<G027.7>
FSCSL<F044.1>
Interrupted
Coordinate established
ZRFx<F120>
CSFO1<F274.4>
FIN<G004.3>
A failure in coordinate establishment prevents the ZRFx signal <F0120> from being set to "1".

Example sequence in which the Cs axis is under composite control in a 2-path system (T series (2-path system))



(1) In path 1, issuing an M code causes the MF signal <F0007.0> to be set to "1".
(2) Make sure that the Cs axis origin established state signal CSPENA <F1048.4> in path 2 is "1", and set the Cs axis coordinate establishment request signal CSFI1<G1274.4> and Cs axis contour control switching signal CON<G1027.7> in path 2 to "1".
(3) The absolute and relative coordinates in path 1 and the machine coordinates in path 2 are established.
(4) Upon completion of count-up, the reference position establishment signal ZRFx<F0120> in path 2 becomes "1". So, reset the Cs axis coordinate establishment request signal CSFI1<G1274.4> in path 2 to "0".
(5) Perform FIN processing for the M code in path 1.

Note

- CMR (parameter No.1820)

Set the command multiplier CMR (parameter No.1820) for the Cs axis to be subjected to composite control to the same value. If it is set to a different value, coordinate establishment does not end normally.

- Synchronous control (T series)

Setting the Cs axis establishment request signal CSFI1<Gn274.4> to "1" during synchronization control results in alarm PS5346.
After disabling synchronization control, perform Cs coordinate establishment, and then start synchronization control again.

- Composite control (T series)

Disabling or enabling composite control during Cs axis coordinate establishment results in alarm PS5346.

- Superimposed Control (T series)

Setting the Cs axis coordinate establishment request signal CSFI1<Gn274.4> to "1" during superimposed control results in alarm PS5346.
After disabling superimposed control, perform Cs coordinate establishment, and then start superimposed control again.

- Simple spindle synchronous control (M series)

If the master axis is parked, coordinate establishment is performed according to the slave axis machine position.
Set the Cs axis coordinate establishment request signal CSFI1<Gn274.4> for the master axis to "1".
The Cs axis coordinate establishment alarm signal CSFO1<Fn274.4> that is output corresponds to the master axis.

- External machine coordinate system shift

Do not shift external machine coordinate systems during Cs axis coordinate establishment.

Signal

Cs axis coordinate establishment request signals CSFI1 <Gn274.4>

[Classification] Input signal
[Function] These signals perform the Cs axis coordinate establishment (Cs contour control function).
[Operation] When the signals become "1", the control unit behaves as follows:
If the signals are "1" in the Cs contour control mode, the absolute and machine coordinates are established according to the Cs axis machine position.
When the signals become "0", the control unit behaves as follows:
The Cs axis coordinate establishment alarm signal CSFO1<F274.4> becomes "0".

Cs axis coordinate establishment alarm signals CSFO1 <Fn274.4>

[Classification] Output signal
[Function] These signals indicate that Cs axis coordinate establishment has not been completed normally.
[Output cond.] These signal is 1 in the following case :
- Cs axis coordinate establishment has not been completed normally.

These signal is 0 in the following case :
- The Cs contour control mode is exited, or
- The corresponding Cs axis coordinate establishment request signal CSFI1<G274.4> becomes "0".

Cs axis origin established state signals First spindle: CSPENA<Fn048.4>

[Classification] Output signal
[Function] These signals indicate that Cs coordinate establishment function is available.
[Output cond.] These signal is 1 in the following case :
- A reference position return is completed normally in the Cs contour control mode.

These signal is 0 in the following case :
- A spindle alarm is detected or the spindle motor rotation speed exceeds the maximum allowable rotation speed (specified in parameter No. 4020).

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn274				CSFI1				
Fn048				CSPENA				
Fn274				CSFO1				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1005								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.

	#7	#6	#5	#4	#3	#2	#1	#0
3700							NRF	

[Input type] Parameter input
[Data type] Bit path

#1 NRF With the first move command (G00) after switching the serial spindle to Cs contour control axis:

0: A reference position return operation is once performed then positioning is performed.
1: A normal positioning operation is performed.

NOTE
1 When using the Cs axis establishment function, this parameter is recommended to be set to 1.
2 The setting of this parameter is valid for G00. The first rapid traverse of a canned cycle is normal positioning regardless of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3712						CSF		

[Input type] Parameter input
[Data type] Bit

#2 CSF In the Cs contour control mode, the function for setting machine coordinates and absolute coordinates based on the machine position of the spindle if the origin is already set up is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
4353			CSPTRE					

[Data type] Bit spindle

#5 CSPTRE Specifies whether to enable Cs axis positional data transfer, as follows:
0: Disable.
1: Enable.

To use this function, reset bit 7 (RFCHK3) of parameter No.4016 to "0".

Alarm and message

Number	Message	Description
PS5346	REFERENCE POSITION RETURN IS NOT PERFORMED	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 CSPENA 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 (T series) for the Cs axis being subjected to coordinate establishment 7. When an attempt is made to start synchronous, composite, or superposition control (T series) for the Cs axis being subjected to coordinate establishment.

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B-64304EN)	Cs contour control
CONNECTION MANUAL (FUNCTION) (B-64303EN-1)	Cs contour control

10.12 MULTI-SPINDLE CONTROL

Overview

In addition to the first spindle, the second spindle can be controlled using an S command from the CNC.
Spindle commands are specified using a single S command as conventionally done. A spindle is selected depending on the signal from the PMC or the address P command.
Gear change between two stages can be made for the second spindle in the same manner as for the first spindle.
A maximum speed can be set for each spindle, and the speed of each spindle can be clamped to the corresponding maximum speed (according to the setting in parameter No. 3772).
The position coder interfaces for the second spindle can be selected and added.
The first to second position coders are selected by signals from the PMC.

NOTE
Multi-spindle control can be used by setting bit 3 (MSP) of parameter No. 8133 to 1 and bit 1 (EMS) of parameter No. 3702 to 0.

Explanation

- Multi-spindle control in machining center

M

- Only when the T-type gear change is selected (when the constant surface speed control function is selected (bit 0 (SCC) of parameter No. 8133 is 1) or when bit 4 (GTT) of parameter No. 3706 is set to 1), multi-spindle control is enabled.
- For information about rigid tapping, see "RIGID TAPPING"

NOTE
For the M series, multi-spindle control is optional function.

- Control methods

There are three types of multi spindle control methods: a method that can use the SIND (spindle motor speed control by the PMC) function only for the first spindle (TYPE-A), a method that can use the SIND function for the two spindles independently (TYPE-B), and a method that is equivalent to TYPE-B and can use address P to select the spindle to be controlled by the S command.

Basic control 1 (Common to TYPE-A and TYPE-B)

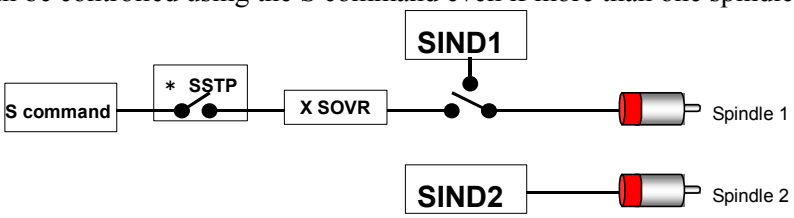
An S command is sent as a speed command to each spindle selected, using a spindle selection signal (SWS1 to SWS2 <Gn027.0 to 1>). Each spindle rotates at the specified speed. If a spindle is not sent a spindle selection signal, it continues to rotate at its previous speed. This allows the spindles to rotate at different speeds at the same time.

Basic control 2 (Common to TYPE-A, TYPE-B, and the method of making spindle selection by address P)

Each spindle also has a spindle stop signal (*SSTP1 to *SSTP2 <Gn027.3 to 4>) to stop its rotation; an unused spindle can still be stopped.
There is a spindle enable signal to control each spindle; ENB <Fn001.4> controls the first spindle, while ENB2<Fn038.2> control the second and third spindles, respectively.
The PMC signals PC2SLC <Gn028.7> is used to select between the first to fourth position coders.

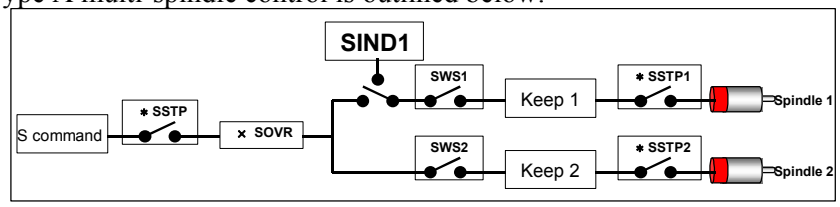
When multi spindle control is disabled

When bit 3 (MSP) of parameter No. 8133 is set to 0, or when bit 3 (MSP) of parameter No. 8133 is set to 1 and bit 1 (EMS) of parameter No. 3702 is set to 1, multi-spindle control is disabled. In this case, only the first spindle can be controlled using the S command even if more than one spindle is present.



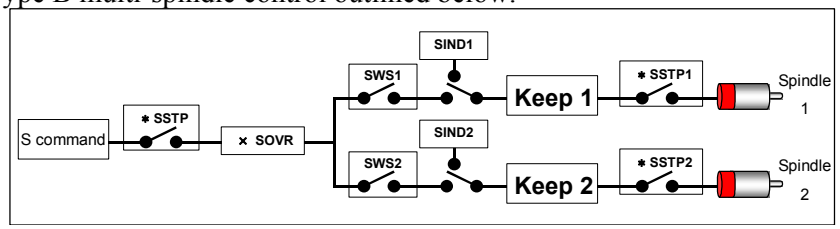
Multi-spindle control (TYPE-A)

When multi-spindle control is enabled (bit 3 (MSP) of parameter No. 8133 is set to 1, and bit 2 (EMS) of parameter No. 3702 is set to 0), TYPE-A is assumed if bit 2 (MSI) of parameter No. 3709 is set to 0.
When the first spindle is selected with the SWS1 signal, the SIND signal <Gn033.7> is used to determine whether the spindle analog voltage is controlled by the PMC or CNC; then signals R01I to R12I <Gn032.0 to Gn033.3> are used to set that spindle's analog voltage. These signals do not affect the second spindle.
The PMC-based polarity (rotation direction) control signals SGN and SSIN <Gn033.5,6> will function for any spindle selected by SWS1 to SWS2.
The concept of Type A multi-spindle control is outlined below.



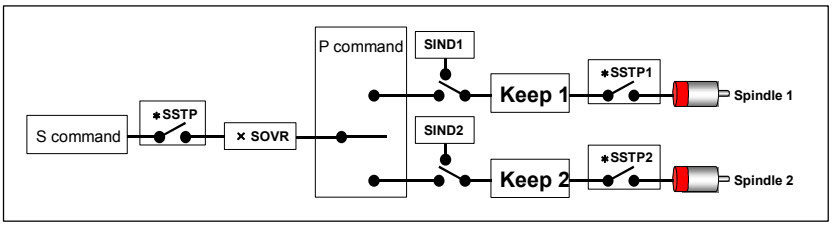
Multi-spindle control (TYPE-B)

When multi-spindle control is enabled (bit 3 (MSP) of parameter No. 8133 is set to 1, and bit 2 (EMS) of parameter No. 3702 is set to 0), TYPE-B is assumed if bit 2 (MSI) of parameter No. 3709 is set to 1.
Each spindle has its own SIND, SSIN and SGN signals. Each of these signals functions regardless of selection state of the spindle selection signals (SWS1 to SWS2).
When either the spindle selection signals (SWS1 to SWS2) or the SIND signal for the first or second spindle is set to "1", the polarity control signals SSIN, SGN will function.
The concept of Type B multi-spindle control outlined below.



Multi spindle control: when a spindle is selected by address P

If bit 3 (MPP) of parameter No. 3703 is set to 1, a spindle is selected by address P. This control method is basically the same as TYPE-B. The first and second spindles each have their own SIND, SSIN, and SGN signals. A spindle is selected by the P command instead of the spindle selection signals (SWS1 to SWS2). The relationship between the P code and the selected spindle is set in parameter No. 3781.
Polarity (rotation direction) control signals SSIN and SGN for each spindle are valid only for the spindle selected by the P command or for the spindle of which SIND signal is 1.
The concept of multi spindle control (spindle selection by address P) is outlined below.



In case of 2-path control, the P code becomes valid in all paths. For example, suppose that the P code to select the first spindle of path 2 is set to 21. When the following is specified in path 1:

S1000 P21;

Then, 1000 is specified for the first spindle in path 2. Therefore, the same P code cannot be used even in different paths.

Override for individual spindles

Different override values can be applied to individual axes separately at a time. The override values can be selected from between 0% and 254%. This function is made usable by setting bit 3 (MSC) of parameter No. 3713 to "1" for multi-spindle control.

- Specification

When each spindle speed override function (parameter EOV(No.3713#4) =1) is effective, the specifications is as following.

- Override signals for each spindle are following.
  SOV0 to 7: Override signal for the first spindle
  SOV20 to 27: Override signal for the second spindle
- Because override is applied to individual axes separately, it is possible to use different values for each axis simultaneously.

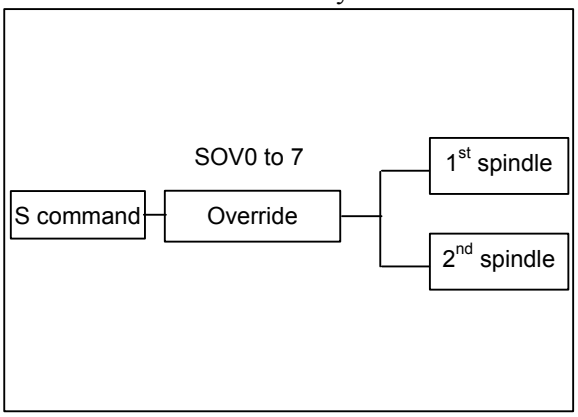


Fig. 10.12 (a) Bit 4 (EOV) of parameter No.3713 = "0"

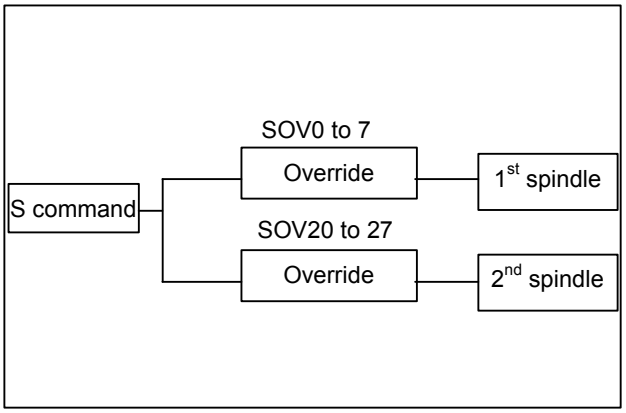


Fig. 10.12 (b) Bit 4 (EOV) of parameter No. 3713 = "1"

Also for spindle control in a 2-path system, this function can be used to apply override to individual axes separately. Shown below is an example of using this function for a 2-path system. In this example, both path 1 and path 2 spindles are controlled by the spindle commands in path 2.

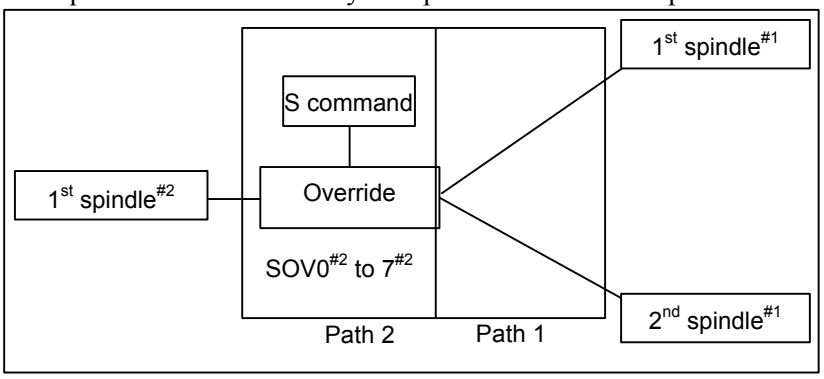


Fig. 10.12 (c) Bit 4 (EOV) of parameter No. 3713 = "0" (2-path system)

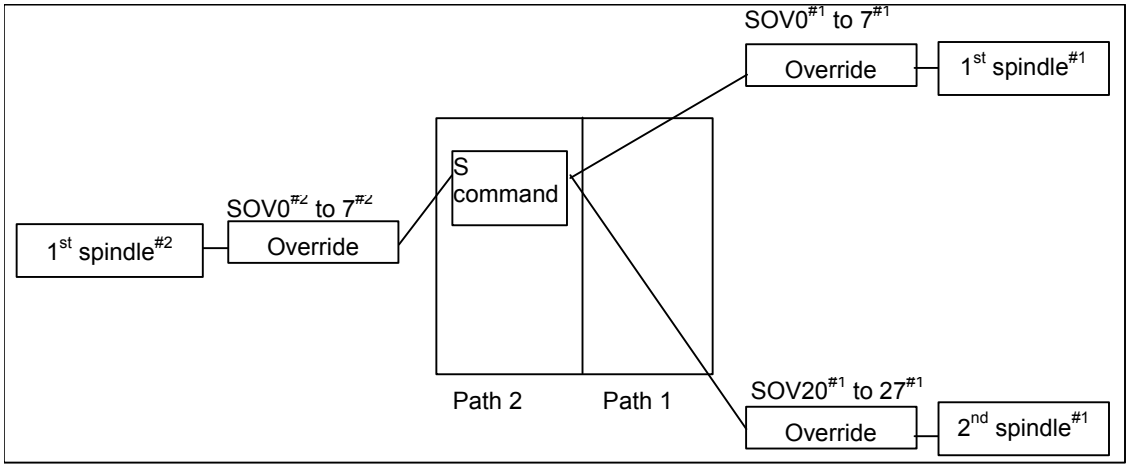


Fig. 10.12 (d) Bit 4 (EOV) of parameter No. 3713 = "1" (2-path system)

- Note

- Before this function can be used, multi-spindle control must be enabled.
  - When constant surface speed control is used, override is applied after the spindle rotation speed is clamped to the clamp speed specified by the spindle rotation speed clamp command (G50S_). Note, therefore, that if an override of 100% or higher is applied, the spindle may rotate faster than the specified clamp speed.
    Bit 5 (SOC) of parameter No. 3708 (for specifying to clamp the spindle rotation speed for constant surface speed control after spindle speed override is put into effect) is invalid. The note given above still applies even if this parameter is set. The spindle rotation speed can be clamped to the clamp speed specified as the spindle rotation speed upper limit (parameter No. 3772).
  - To display the spindle override value, set bit 5 (SOV) of parameter No. 3106 to 1. (Bit 5 (SOV) of parameter No. 3106 is valid when bit 2 (DPS) of parameter No. 3105 is set to 1.)

NOTE
Multi-spindle control can be used by setting bit 3 (MSP) of parameter No. 8133 to 1 and bit 2 (EMS) of parameter No. 3702 to 0.

- When this function is in use, a spindle override of 100% is applied during tapping cycle mode operation for the T series, G84 and G88) or threading operation (for the T series, G32, G92, and G76). Setting bit 6 (TSO) of parameter No. 3708 to "1" enables spindle override.
  - Parameter No. 5131 (for setting a threading (chamfering) angle) is unusable. Keep the parameter (No. 5131) reset to 0.
  - For spindles not subjected to override, keep their spindle override signals set to 100%.
  - No spindle override is applied to spindles which are under spindle rotation speed control directed by the PMC.

Position coder selection by address P (bit 6 (MPC) of parameter No. 3713 is set to 1)

In multi-spindle control, when the spindle is to be selected using address P in a program (bit 3 (MPP) of parameter No. 3703 is set to 1), position coder feedback used for operations such as thread cutting and feed per revolution can also be switched automatically according to the selected spindle by setting bit 6 (MPC) of parameter No. 3713 to 1.

- Explanation

In multi-spindle control, when address P is used to select a spindle (bit 3 (MPP) of parameter No. 3703 is set to 1), address P specified as follows is assumed to be a command for spindle selection:

S_ P_ ;

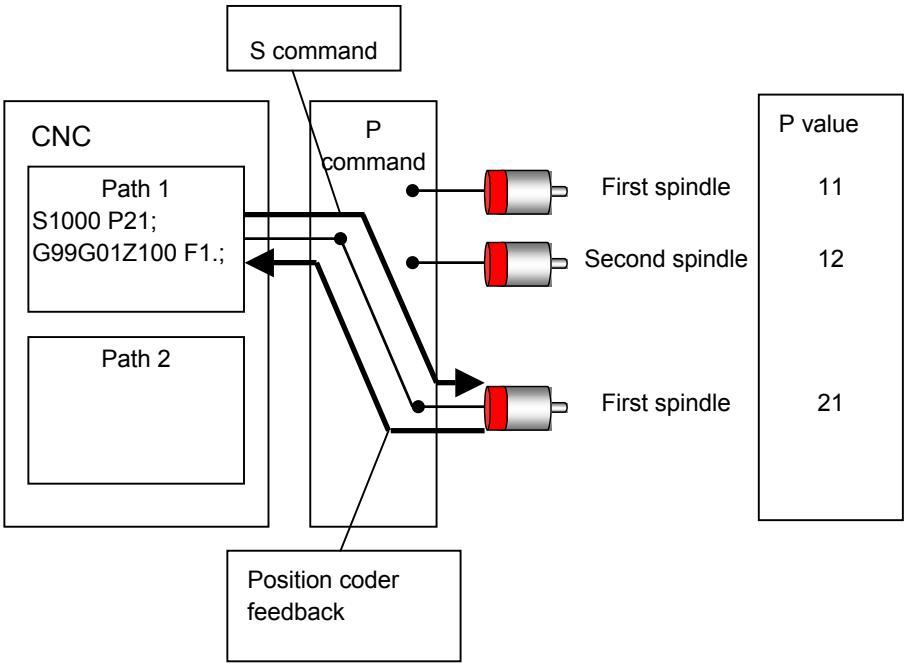
If the above is specified when bit 6 (MPC) of parameter No. 3713 is set to 1, the position coder of the selected spindle is automatically selected as if the position coder were selected by using position coder select signal PC2SLC<Gn028.7> and inter-path spindle feedback signals SLPCA<Gn064.2> and SLPCB<Gn064.3>. (In this case, these signals are ignored even when they are set from the PMC ladder.)

- Example

In the following program:

```
S1000 P21 ;
G99 G01 Z100. F1. ;
```

(1) The first spindle in path 2 is specified by S1000.
(2) Position coder feed-back for the first spindle in path 2 is selected automatically.
(Used in feed per revolution)



- Relationship with other optional functions

Constant surface speed control

The control function for keeping the surface speed constant can be used with any of the two spindles, if the spindle speed is within the range allowable for this function. (If position coders are required, this function is enabled for serial spindles.) The spindle selection signal (SWS1 to SWS2) for the spindle must stay set at "1" during machining using this function.

Spindle speed fluctuation detection

When this function is used with multi spindle control, multiple position coders can be used. So, when a spindle speed fluctuation is detected, attention must be paid to the states of the position coder selection signal (PC2SLC) and spindle selection signals (SWS1 to SWS2)

Actual spindle speed output

The actual spindle speed output function conveys speed information obtained from the position coder specified by the position coder selection signal (PC2SLC) to the PMC.

Polygon turning

Polygon turning rotates a tool axis in phase with the spindle. To perform polygon turning when multi-spindle control issued, select the spindle and the position coder associated with the spindle.

Spindle synchronization, polygon turning between spindles

During spindle synchronization or polygon turning between spindles, the slave spindle operates in phase with the master spindle. Multi-spindle control for the master spindle can be used during synchronization control, but multi-spindle control for the slave spindle is disabled.

Rigid tapping

One of the first to second spindles can be selected as the rigid tapping axis according to the states of the spindle selection signals (SWS1 to SWS2) to perform rigid tapping. However, the following restrictions are imposed:

- Set the SWS1 to SWS2 signals before directing rigid tapping;
- Do not switch the SWS1 to SWS2 signals during rigid tapping; and
- Use the appropriate ENB signal (either ENB or ENB2) for the selected spindle as the ENB signal for the rigid tapping PMC sequence.

The spindles not used for rigid tapping can be rotated at a speed specified before rigid tapping starts, or can be stopped.

Signal

Spindle Selection Signals SWS1,SWS2<Gn027.0,.1>

[Classification] Input signal

[Function] Controls whether S command specified to the NC is output to the spindle or not in multi-spindle.

SWS1 1 : Outputs a speed command to the first spindle.
0 : Outputs no speed command to the first spindle.

SWS2 1 : Outputs a speed command to the second spindle.
0 : Outputs no speed command to the second spindle.

Individual spindle stop signals *SSTP1,*SSTP2<Gn027.3,.4>

[Classification] Input signal

[Function] Effective only to multi-spindle, each spindle can be stopped by this signal.

*SSTP1 1 : Does not set 0 min^{-1} for output to the first spindle.
0 : Sets 0 min^{-1} for output to the first spindle.

*SSTP2 1 : Does not set 0 min^{-1} for output to the second spindle.
0 : Sets 0 min^{-1} for output to the second spindle.

Gear select signals GR11<Gn028.1>,GR12<Gn028.2>,GR21<Gn029.0>,GR22<Gn029.1>

[Classification] Input signal

[Function] Gear selection signals when multi-spindle is equipped. These signals post the gear stage currently selected (4 stages).

Gear	GRs1	GRs2
Gear 1	0	0
Gear 2	1	0
Gear 3	0	1
Gear 4	1	1

(s is 1, 2.)

Position coder selection signal PC2SLC<Gn028.7>

[Classification] Input signal

[Function] These signals select the position coder of the serial spindle used for multi-spindle control.

Position coder to be selected	PC2SLC
First position coder	0
Second position coder	1
Third position coder	0
Fourth position coder	0

When the second position coder is not installed, do not switch this signal. Always set "0" for first position coder.

NOTE
If a spindle number that is not a valid spindle number is selected, or if the specified spindle number belongs to a path that has no valid spindle, alarm SP1202 is issued.

Spindle enable signals ENB<Fn001.4>, ENB2<Fn038.2>

[Classification] Output signal

[Function] These signals inform PMC of whether or not to perform output to the second spindle in multi-spindle control. For the first spindle, ENB <Fn001.4> can be used as usual. The signals are used as a condition to stop the analog spindle, and are also used for a PMC ladder sequence that is associated with rigid tapping. (See "RIGID TAPPING.")

[Output cond.] ENB Enables output of command other than 0 to the first spindle control unit.
Outputs 0 to the first spindle control unit.

ENB2 Enables output of command other than 0 to the second spindle control unit.
Outputs 0 to the second spindle control unit.

Spindles control signal by PMC
For first spindle SIND,SSIN,SGN<Gn033.7,6,5>,R01I to R12I<Gn032.0 to Gn033.3>
For second spindle SIND2,SSIN2,SGN2<Gn035.7,6,5>,R01I2 to R12I2<Gn034.0 to Gn035.3>

[Classification] Input signal

[Function] The spindle motor of each spindle can be controlled by issuing commands from the PMC. The speed command and polarity (rotation direction) of the spindle motor can be controlled. Usually, CNC commands are used to specify a speed and polarity. By using these signals, whether commands issued from the CNC or PMC are to be used for this control can be selected.

Even when multi-spindle control is not being used, the signals can be used to control the second spindle.

When multi-spindle control is being used, and TYPE-A is selected (bit 2 (MSI) of parameter No. 3709 is set to 0), the signals for the second spindle cannot be used.

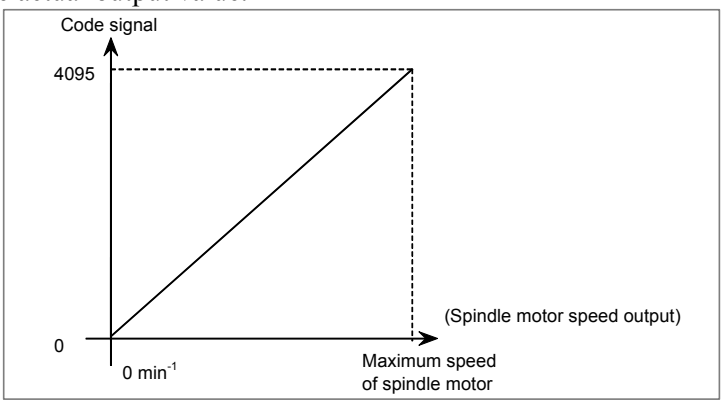
For details of each signal, see "SPINDLE OUTPUT CONTROL BY THE PMC."

S12-bit code signals
R01O to R12O<Fn036.0 to Fn037.3>, R01O2 to R12O2<Fn200.0 to Fn201.3>,

[Classification] Output signal

[Function] If bit 3 (MRS) of parameter No. 3709 is 1, the spindle speed command value calculated by the CNC is converted to a code signal from 0 to 4095, and the conversion result is output to a spindle. If MRS is set to 0, the data of the currently selected spindle (when more than one spindle is selected, the spindle with the smallest number) is output only to R01O to R12O for the first spindle.

[Output cond.] The relationship between the spindle speed command output value (the value calculated by the CNC) and the value output to these signals is plotted in the graph shown below. Note that for either a serial or analog spindle, the output value on these signals is a value 0 to 4095 as a result of conversion from the spindle speed command output value calculated by the spindle control function of the CNC; the output value on these signals is not the actual output value.



Address P signals MSP00 to MSP15<Fn160.0 to Fn161.7>

[Classification] Output signal

[Function] The P value specified last by the S_P_; command is output.

[Output cond.] When multi spindle control by using address P is enabled (bit 3 (MPP) of parameter No. 3703 is 1), the P value specified in the S_P_; command is output. When no S_P_; command has been specified even once since power-up, the initial P value set in parameter No. 3775 is output.

1st spindle speed override signal SOV0 to SOV7<Gn030>
2nd spindle speed override signal SOV20 to SOV27<Gn376>

[Classification] Input signal

[Function] If bit 4 (EOV) of parameter No. 3713 = 1, an override of 0% to 254% of the rotation speed specified by the CNC can be applied to individual spindles separately.

[Operation] Specify an override value, using an 8-bit binary number. If all bits are "1", however, an override of 0% is assumed.

NOTE
If bit 4 (EOV) of parameter No. 3713 = "0", the override signals SOV0 to SOV7 are applied to all spindles that receive speed commands.
The signals SOV20 to SOV27 are invalid.

Spindle command path specification signal SPSP<Gn536.7>

[Classification] Input signal

[Function] Turns spindle command path specification ON or OFF.

[Operation] If a command based on address P (bit 3 (MPP) of parameter No.3703 = "1") is used in multi-spindle control:

- Setting this signal to "1" enables path specification for spindle commands. Spindle commands can be issued from the path specified by parameter No. 11090. If a spindle command is issued from a path other than the path, PS5305 alarm is issued.
- Setting this signal to "0" disables path specification for spindle commands.

Spindle commands can be issued from individual paths no matter what the setting of parameter No. 11090.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn027				*SSTP2	*SSTP1		SWS2	SWS1
Gn028	PC2SLC					GR2	GR1	
Gn029		*SSTP					GR22	GR21
Gn030	SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0
Gn032	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
Gn033	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
Gn034	R08I2	R07I2	R06I2	R05I2	R04I2	R03I2	R02I2	R01I2
Gn035	SIND2	SSIN2	SGN2		R12I2	R11I2	R10I2	R09I2
Gn376	SOV27	SOV26	SOV25	SOV24	SOV23	SOV22	SOV21	SOV20
Gn536	SPSP							

	#7	#6	#5	#4	#3	#2	#1	#0
Fn001				ENB				
Fn036	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
Fn037					R12O	R11O	R10O	R09O
Fn038						ENB2		
Fn160	MSP07	MSP06	MSP05	MSP04	MSP03	MSP02	MSP01	MSP00
Fn161	MSP15	MSP14	MSP13	MSP12	MSP11	MSP10	MSP09	MSP08
Fn200	R08O2	R07O2	R06O2	R05O2	R04O2	R03O2	R02O2	R01O2
Fn201					R12O2	R11O2	R10O2	R09O2

Parameter

Parameters related to the first spindle and first position coder are the same as those used conventionally. The following explains the parameters added in connection with this function and related parameters:

	#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.

NOTE
Make the setting on the side of the path in which multi-spindle control is unnecessary in 2-path control.

	#7	#6	#5	#4	#3	#2	#1	#0
3703					MPP			

[Input type] Parameter input
[Data type] Bit

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#3 MPP In multi-spindle control, a spindle selection using a programmed command instead of using the signals (SWS1 to SWS2<Gn027.0 to 1>) is:
0: Not made.
1: Made.

NOTE
1 When this parameter is set to 1, set parameter No. 3781 at the same time.
2 If this parameter is set to 1, position coder feedback used for operations such as thread cutting and feed per revolution can also be switched automatically according to the selected spindle by setting bit 6 (MPC) of parameter No. 3713 to 1. (Position coder selection by address P)

	#7	#6	#5	#4	#3	#2	#1	#0
3706						MPA		
				GTT		MPA		

[Input type] Parameter input
[Data type] Bit path

#2 MPA If a spindle is to be selected using a P command (with bit 3 (MPP) of parameter No. 3703 set to 1) in multi-spindle control, and a P command is not specified together with an S command:
0: The alarm (PS5305) is issued.
1: The last P specified by S_ P_; (by S_ P_; specified for the path in case of a 2-path system) is used. If P is not specified even once after power-up, the value of parameter No. 3775 is used.

NOTE
This parameter is valid only when bit 3 (MPP) of parameter No. 3703 is set to 1.

#4 GTT Selection of a spindle gear selection method
0: Type M.
1: Type T.

NOTE
1 M type
The gear selection signal is not input. The CNC selects a gear based on the speed range of each gear set by a parameter beforehand according to S codes, and the selected gear is posted by outputting the gear selection signal. Moreover, the spindle speed matching the gear selected by the output gear selection signal is output.
T type
The gear selection signal is input. The spindle speed matching the gear selected by this signal is output.
2 When constant surface speed control is selected (bit 0 (SSC) of parameter No. 8133 is 1), the T type is assumed regardless of the setting of this parameter.
3 When type T spindle gear switching is selected, the following parameters have no effect:
No.3705#2(SGB), No.3751, No.3752,
No.3705#1(GST), No.3705#3(SGT), No.3761, No.3762,
No.3705#6(SFA), No.3735, No.3736
On the other hand, parameter No. 3744 becomes usable.

	#7	#6	#5	#4	#3	#2	#1	#0
3708		TSO						

[Input type] Parameter input
[Data type] Bit path

#6 TSO During a threading or tapping cycle, the spindle override is:
0: Disabled (tied to 100%).
1: Enabled.

NOTE
During rigid tapping, the override is tied to 100%, irrespective of the setting of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3709					MRS	MSI		

[Input type] Parameter input
[Data type] Bit path

#2 MSI In multi-spindle control, the SIND signal is valid
0: Only when the first spindle is valid (SIND signal for the 2nd, 3rd spindle becomes ineffective) (TYPE-A)
1: For each spindle irrespective of whether the spindle is selected (Each spindle has its own SIND signal). (TYPE-B)

#3 MRS When the actual spindle speed signals and S 12-bit code signals are output in multi-spindle control:

0: The signals common to the first spindle and second spindle are used, and the signals for the spindle selected by the spindle selection signal are output.

1: The signals for the first spindle and the signals for the second spindle are output separately.

> **NOTE**
> This parameter requires multi-spindle control (bit 3 (MSP) of parameter No. 8133 is 1) and spindle serial output (bit 5 (SSN) of parameter No. 8133 is 0).

	#7	#6	#5	#4	#3	#2	#1	#0
3713		MPC		EOV	MSC			

[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.

#3 MSC Multi-spindle control TYPE-C is:

0: Not used.

1: Used.

> **NOTE**
> If parameter MSC and MSI (bit 2 of parameter No. 3709 for multi-spindle control TYPE-B) are set to 1 at the same time, multi-spindle control TYPE-C is enabled.

#4 EOV Each spindle speed override is:

0: Not used.

1: Used.

> **NOTE**
> Multi-spindle control TYPE-C (No.3713#3=1) is necessary to use this function.

#6 MPC When a spindle is selected with address P in a program during multi-spindle control (bit 3 (MPP) of parameter No. 3703 is set to 1), position coder feedback used for thread cutting, feed per revolution, and so forth is:

0: Not changed automatically according to the selected spindle.

1: Changed automatically according to the selected spindle.

> **NOTE**
> Setting this parameter produces the same effects as when position coder select signal PC2SLC<Gn028.7>, inter-path spindle feedback signals SLPCA<Gn064.2> and SLPCB<Gn064.3> are set.
> At this time, even when an attempt to set these signals is made by a PMC ladder, these signal operations are ignored.

3720	Number of position coder pulses

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] Detection unit
[Valid data range] 1 to 32767
Set the number of position coder pulses.

3721	Number of gear teeth on the position coder side

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 9999
Set the number of gear teeth on the position coder side in speed control (such as feed per revolution and threading).

3722	Number of gear teeth on the spindle side

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 0 to 9999
Set the number of gear teeth on the spindle side in speed control (such as feed per revolution and threading).

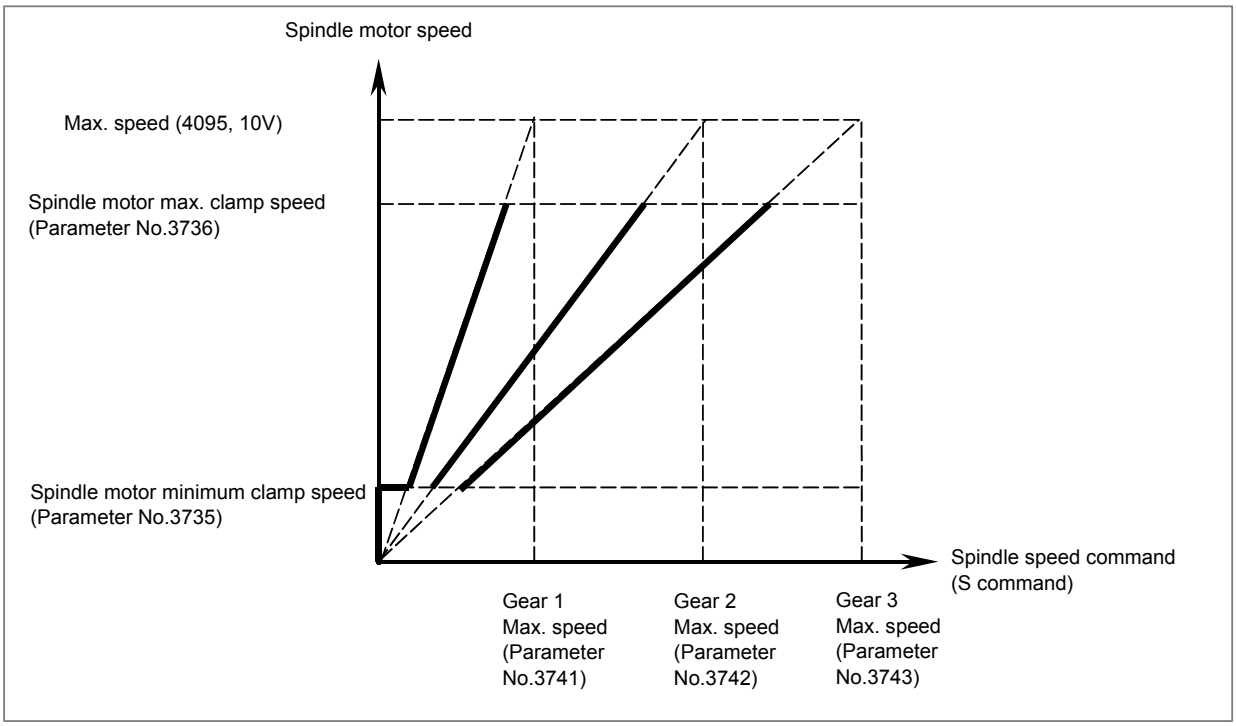
3741	Maximum spindle speed for gear 1

3742	Maximum spindle speed for gear 2

3743	Maximum spindle speed for gear 3

3744	Maximum spindle speed for gear 4
	(Note)

[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.



NOTE
If a type-T gear shift scheme is selected for the M series (with the constant surface speed control option installed or bit 4 (GTT) of parameter No. 3706 = 1), parameter No. 3744 is usable also in the M series.
Note, however, that, even in this case, only up to three main gear stages are usable for rigid tapping.

3772	Maximum spindle speed

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999

This parameter sets the maximum spindle speed.
When a command specifying a speed exceeding the maximum speed of the spindle is specified , or the speed of the spindle exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

⚠ CAUTION
1 When 0 is set in this parameter, the speed of the spindle is not clamped.
2 When spindle speed command control is applied using the PMC, this parameter has no effect, and the spindle speed is not clamped.

NOTE
1 For the M series, this parameter is valid when constant surface speed control is selected (bit 0 (SSC) of parameter No. 8133 is 1).

NOTE
2 When the constant surface speed control is selected, the spindle speed is clamped at the maximum speed, regardless of whether the G96 mode or G97 mode is specified.

3775 **Default P command value for spindle selection in multi-spindle control**

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 32767
When bit 3 (MPP) of parameter No. 3703 is set to 1 and bit 2 (MPA) of parameter No. 3706 is set to 1 in multi-spindle control, set a default P command value applicable if S_P_ is not specified even once after power-up.

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 2,
S1000 P2;
causes the second spindle to rotate at S1000.

NOTE
1 This parameter is valid if bit 3 (MPP) of parameter No. 3703 is set to 1.
2 If this parameter is set to 0, the corresponding spindle cannot be selected by a P code.
3 Under 2-path control, the P code specified here is valid for each path.
For instance, if the P code to select the first spindle of path 2 is set to 21, specifying S1000 P21; in path 1 causes the first spindle of path 2 to be rotated at S1000.
4 Identical P code values cannot be used for different spindles. (Identical P code values cannot be used even if the paths are different.)
5 When this parameter is used (when bit 3 (MPP) of parameter No. 3703 is set to 1), the spindle command selection signal is invalid.
6 To use this parameter, enable multi-spindle control (bit 3 (MSP) of parameter No. 8133 is 1).

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 2
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 2: 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) 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 SWS2 <Gn027.0 to 1>).

	#7	#6	#5	#4	#3	#2	#1	#0
8133					MSP			

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#3 MSP Multi-spindle is:
0: Not Used.
1: Used.

NOTE
For the M series, multi-spindle control is optional function.

Caution

⚠ CAUTION
1 While a function using a position coder feedback such as feed per revolution and a threading command is being executed, avoid changing the selection of a position coder by using position coder selection signal PC2SLC.
2 Type A multi-spindle control differs from Type B in the relationship between the SWS1 and SIND signals for the first spindle. In Type B, SIND functions only when SWS1 is set to "1". In Type A, SIND functions whether SWS1 is "1" or "0"; each spindle is selected by either of its respective SWS1 or SIND signals being set to "1".

Note

NOTE
1 The spindle orientation signal, spindle speed override signals, and spindle stop signal *SSTP only function for the selected signals.
2 The multi-spindle function allows multiple position coder interfaces to be used. But the number of actual speed indications on the CNC screen does not change. The speed based on the feedback information of the selected position coder is displayed.

NOTE
3 An SOR command has priority over S commands and SIND-based rotation control from the PMC, and will cause all selected spindles to perform orientation rotation.
4 If the position coder select signal selects a spindle number that is not a valid spindle number or a spindle number that belongs to a path having no valid spindle, alarm SP1202 is issued.

Alarm and message

Number	Message	Description
PS0333	TOO MANY SPINDLE COMMANDS	Only one spindle could be commanded in the same block.
PS5305	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 parameter EMS (No. 3702#1) 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.
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.

10.13 RIGID TAPPING

Overview

In a tapping cycle (M series: G84/G74, T series: G84/G88), synchronous control is applied to the tapping operation of a tapping axis and the operation of the spindle.
This capability eliminates the need to use a tool such as a float tapper, thus enabling higher-speed, higher-precision tapping.
Whether a tapping cycle is an ordinary tapping cycle or rigid tapping cycle is determined by the miscellaneous function code for rigid tapping M29. (A different M code can be used by setting the parameters accordingly, but M29 is used in the description given here.)
By setting the parameters, G codes for tapping cycles can be changed to G codes for rigid tapping only. In this case, the CNC specifies M29 internally.
To perform rigid tapping, the following must be added to the ordinary connections:

- Connection of a position coder to the spindles
- Addition of a sequence to the PMC
- Setting of related parameters

To avoid duplicate descriptions, assume the following unless noted otherwise:

G code for a tapping cycle	
M series	**T series**
G84/G74	G84/G88

Gear selection method	
M series	**T series**
M-type or T-type gear selection method	T-type gear selection method only

Parameters used according to the number of gear stages
No.5221 to No.5224, No.5231 to No.5234, No.5241 to No.5244, No.5261 to No.5264, No.5271 to No.5274, No.5281 to No.5284, No.5291 to No.5294, No.5321 to No.5324, etc.

⚠ CAUTION

1 In the text of this section, a description of up to four-step gears is provided. Gears of four steps are used with the T series, but gears of up to only three steps are used with the M series.
2 When M-type gear selection is used for the M series, the maximum spindle speed for rigid tapping (specified with parameters No. 5241 to 5243) must also be set for parameter No. 5243 regardless of the number of gear steps. (For a system having a single gear step, set the same value as that of parameter No. 5241 for parameter No. 5243. For a system having two gear steps, set the same value as that of parameter No. 5242 for parameter No. 5243.)

The descriptions given in this section (such as spindle gear switching and M-type/T-type) are based on the explanation given in Section "SPINDLE SPEED CONTROL."
Refer to Section "SPINDLE SPEED CONTROL" as necessary.

- Specification of M series/T series

The differences in the specifications for rigid tapping for the M series and T series are described below.

M

Rigid tapping of M series

The tapping cycle G84 and the reverse tapping cycle G74 can be used to specify machining center system rigid tapping.
A drilling axis can be arbitrarily selected from the basic axes X, Y, Z, as well as axes parallel to the basic axes, by setting the corresponding parameters accordingly (bit 0 (FXY) of parameter No. 5101).
The spindle operations of G84 and G74 are reversed with respect to each other.

T

Rigid tapping of T series

The face tapping cycle G84 and the side tapping cycle G88 can be used to specify lathe system rigid tapping.
Depending on the rigid tapping command, rigid tapping can be performed along the Z-axis (when G84 is used) or the X-axis (when G88 is used).
A reverse tapping cycle, like that supported by M series, is not available.
When multi-spindle control is provided as well as rigid tapping on the first spindle (either analog or serial spindle), the second spindle (serial) may also be selected for rigid tapping.

See "Rigid Tapping with Spindle of Another Path" for explanations about rigid tapping between different paths (rigid tapping on a spindle for tool post 2 as directed by a command for tool post 1) in a 2-path system.

- Multi spindle control

The relationships between rigid tapping and multi spindle control are described below.
For details of multi-spindle control, see "MULTI-SPINDLE CONTROL" described previously.

Without multi spindle control

Rigid tapping can be performed with the first spindle only.

With multi spindle control

Rigid tapping can be performed by selecting the second spindle in addition to the first spindle.

10.13.1 Connection Among Spindle, Spindle Motor, and Position Coder

As shown in the figure below a gear ratio can be inserted between the spindle and spindle motor (n : m), and between the spindle and position coder (N : M).

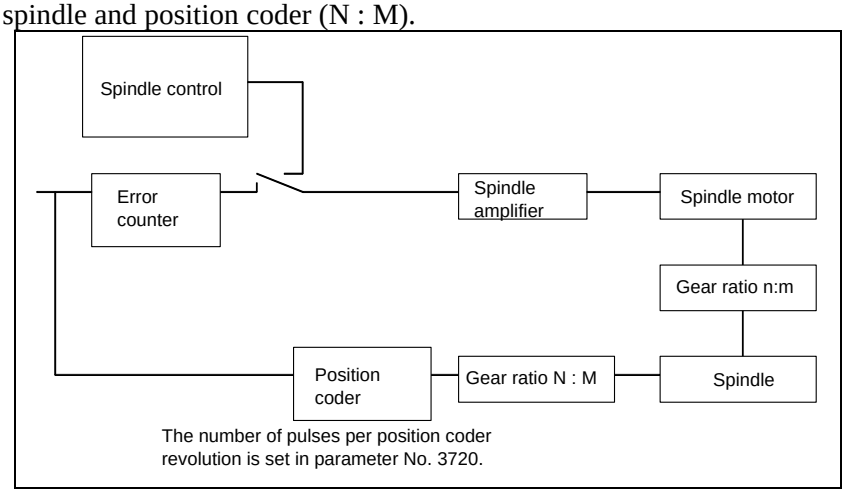


Fig. 10.13.1 (a) When a position coder is externally attached (on the spindle side)

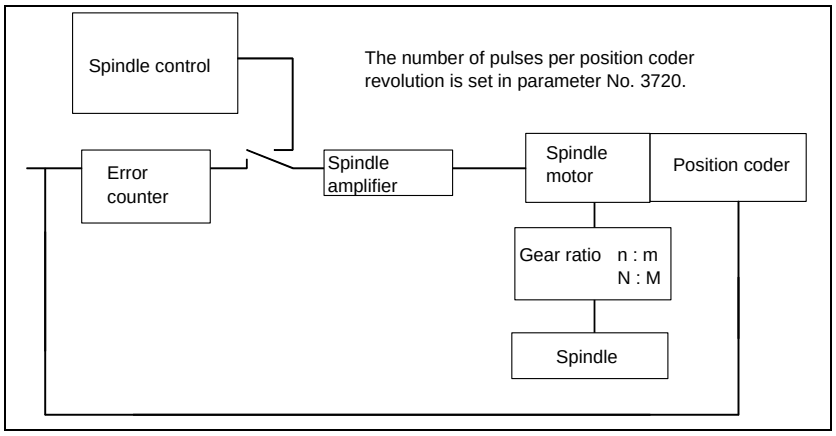


Fig. 10.13.1 (b) When a position coder is built into a spindle motor

(1) Gear between spindle and spindle motor

Up to three gear stages (M series) or four gear stages (T series) can be provided between the spindle and the spindle motor.

The gear ratio is arbitrary. The distance of spindle rotation per revolution of spindle motor is different, based on the gear ratio. The speed command to the spindle motor will be adjusted.

(2) Gear between spindle and position coder

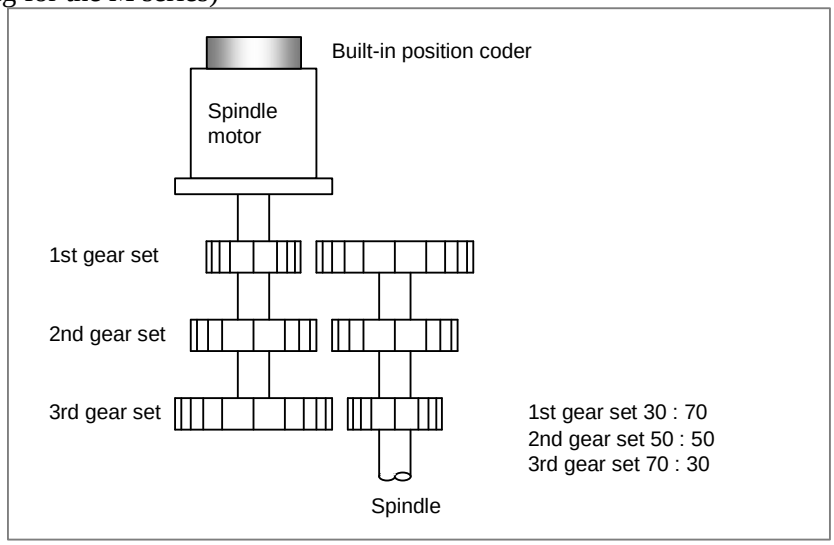
The position coder is used to detect the position of the spindle.

The gear ratio between the spindle and position coder is set in parameter No. 5221 to No. 5224 and No. 5231 to No. 5234.

When a position coder is externally attached, the gear ratio between the spindle and position coder is not changed by a gear change between the spindle motor and spindle. So, set the same number of teeth in parameter No. 5221 to No. 5224 and No. 5231 to No. 5234.

When a position coder is built into a spindle motor, the gear ratio between the spindle and position coder is changed by a gear change between the spindle motor and spindle. So, set the number of teeth for each gear in parameter No. 5221 to No. 5224 and No. 5231 to No. 5234.

Example of setting for the M series)



Parameter No.	Set value	Meaning
5221	70	Number of teeth of the 1st gear for the spindle side
5222	50	Number of teeth of the 2nd gear for the spindle side
5223	30	Number of teeth of the 3rd gear for the spindle side
5231	30	Number of teeth of the 1st gear for the position coder side

Parameter No.	Set value	Meaning
5232	50	Number of teeth of the 2nd gear for the position coder side
5233	70	Number of teeth of the 3rd gear for the position coder side

For the T series, up to four gear stages can be set.

(3) Rigid tapping and machines with multiple gears

When performing rigid tapping on a machine that has multiple gears, note the points below.

- M type gear selection method

M

The CNC determines whether gears need changing using the gear change specification mentioned in section “SPINDLE SPEED CONTROL”. If the gears need to be changed, the CNC generates the S function code read signal SF <F007.2> and gear selection signals GR1O, GR2O, and GR3O <F034.0-2> to notify the PMC. Change gears using the PMC, based on these signals.

- T type gear selection method

The CNC does not process gear changes. Using the PMC, determine whether gears need changing, and make the change if needed.
When the CNC has the S function code, it outputs signal SF <F007.2> and S function code signals S00 to S31 <F022-F025> to the PMC. (However, parameter No. 3705 and its related parameters need to be set for S code and SF signal output).
The PMC side finds a gear whose spindle speed range includes the spindle speed specified by an S function code. A selected gear is to be output onto the gear selection signals (G028.1,2 for the first spindle and G029.0,1 for the second spindle) for notification to the CNC. Regardless of the gear selection method, the clutch/gear selection signals CTH2 and CHT1 (G070.3,2 for the first spindle and G074.3,2 for the second spindle) corresponding to a gear change are to be input for notification through the CNC to the serial spindle control unit.

Changing gears during rigid tapping requires a different process from that for gear changes during normal machining. As described above, changing gears conforms to the gear change specifications mentioned in section "SPINDLE SPEED CONTROL" when the M type gear selection method has been selected. With the T type gear selection method, changing gears conforms to the logic programmed in the PMC.
Regardless of the option's selection, if the range in which the spindle speed specified by the S function code does not correspond to the currently selected gear, the gears are changed.
The following tables list the spindle speed ranges for each gear during normal machining (assuming no machine restrictions) and rigid tapping:

Gear	Spindle speed range (normal machining)	
	Lower limit	Upper limit
Low-speed gear	1 revolution	Maximum low-speed gear speed = $\frac{\text{Maximum spindle motor speed} \times L\%}{\text{Low-speed gear ratio}}$
Medium speed gear	Maximum low-speed gear speed + 1 revolution	Maximum medium-speed gear speed = $\frac{\text{Maximum spindle motor speed} \times L\%}{\text{Medium speed gear ratio}}$
High-speed gear	Maximum medium-speed gear speed + 1 revolution	Maximum high-speed gear speed = $\frac{\text{Maximum spindle motor speed} \times L\%}{\text{High-speed gear ratio}}$

NOTE
This table shows an example of three gears. L% indicates a spindle motor protection constant (up to 100). L can be specified for each gear using method B for changing in M type gear selection method (bit 2 (SGB) of parameter No. 3705 is set to 1).

The spindle speed range of each gear in rigid tapping must satisfy the table below.

Gear	Spindle speed range (during rigid tapping)	
	Lower limit	Upper limit
Low-speed gear	1 revolution	Maximum low-speed gear speed $= \frac{\text{Basic spindle motor speed} + \alpha}{\text{Low-speed gear ratio}}$
Medium speed gear	Maximum low-speed gear speed + 1 revolution	Maximum medium-speed gear speed $= \frac{\text{Basic spindle motor speed} + \alpha}{\text{Medium-speed gear ratio}}$
High-speed gear	Maximum medium-speed gear speed + 1 revolution	Maximum high-speed gear speed $= \frac{\text{Basic spindle motor speed} + \alpha}{\text{High-speed gear ratio}}$

NOTE
This table show an example of three gears. For the basic spindle motor speed, refer to the spindle motor description manual. "+α" means that the spindle motor speed may slightly exceed the basic spindle motor speed.

If the M type gear selection method is used, use gear change method B (bit 3 (SGT) of parameter No. 3705 is set to 1) in the tapping cycle to specify the following:
The table above shows the maximum low-speed gear speed during rigid tapping for low-/medium-speed gear change position D (parameter No. 3761). The table above shows the maximum medium-speed gear speed during rigid tapping for medium-/high-speed gear change position E (parameter No. 3762). If the T type gear selection method is used, add the rigid tapping logic to the logic programmed in the PMC. See Section "SPINDLE SPEED CONTROL" for details of the spindle gear change specifications. The loop gain can be specified for each gear.
In parameter Nos. 5281 to 5284, set the loop gain of each gear. Set a loop gain multiplier in parameter Nos. 5291 to 5294. Specify the time constant and the maximum spindle speed for each gear.
Use parameters Nos. 5261 to 5264 to specify the time constant.
Use parameters Nos. 5241 to 5244 to specify the maximum spindle speed.
For M type gear selection method, set the maximum spindle speed to parameter No. 5243, irrespective of the number of gear stages used.
Setting bit 2 (TDR) of parameter No. 5201 to "1" enables setting of the time constants used during extraction for each gear set. Specify the extraction time constant for each gear in parameter Nos. 5271 to 5274.
It is recommended that a value of about 1:8 to 8:1 be actually used as the gear ratio between the spindle and position coder.

10.13.2 Rigid Tapping Specification

- Feed rate

In rigid tapping mode, the drilling axis is fed at a rate specified by F. The spindle speed is specified by S 360 [deg/min].
A detailed description of commands for feed per minute and feed per revolution is provided later.

- Acceleration/deceleration after interpolation

Linear acceleration/deceleration or bell-shaped acceleration/
deceleration can be applied.
A detailed description is provided later.

- Acceleration/deceleration before look-ahead interpolation

Acceleration/deceleration before look-ahead interpolation is disabled.

- Override

Various types of overrides are disabled. However, the following
overrides can be enabled by setting parameters:

- Extraction override
- Override signal

A detailed description is provided later.

- Dry run

Dry run is valid for G84/G74 (for M series) or G84/G88 (for T series). When the dry run is applied to the drilling axis speed of G84/G74 or G84/G88, tapping is performed. The spindle speed will match the dry run speed.

- Machine lock

When G84/G74 (for M series) or G84/G88 (for T series) is executed with the machine locked, however the drilling axis and the spindle do not move.

- Reset

When the reset operation is performed during rigid tapping, the mode is reset. The spindle motor goes to the ordinary mode. However, the G84/G74 (for M series) or G84/G88 (for T series) mode may not be canceled with bit 6 (CLR) of parameter No. 3402.

- Feed hold and single block

In the G84/G74 (for M series) or G84/G88 (for T series) mode, the feed hold and single block functions are disabled when bit 6 (FHD) of parameter No. 5200 is set to 0. The functions are enabled when the bit is set to 1.

- Operation mode

G84/G74 (for M series) or G84/G88 (for T series) can be executed only in the MEM and MDI modes.

- Manual feed

It is impossible to perform rigid tapping by manual feed.

- Backlash compensation

In rigid tapping mode, the backlash is compensated for the lost motion at forward and reverse spindle rotations. Set it using parameter No. 5321 to No 5324. The normal backlash compensation is inserted on the drilling axis.

- C-axis clamp/unclamp

T

An M code can be specified to clamp/unclamp the C-axis mechanically in rigid tapping. By adding an M code for clamping to a block specifying G84/G88 (for T series), both M codes are output. A description of timing is provided later.
An M code for clamping is to be set in parameter No. 5110. An M code for unclamping assumes the value set in parameter No. 5110 + 1.

M code for C-axis clamping No.5110	
0	**Other than 0**
No M code is output.	M code for unclamping is the setting of No. 5110 plus 1.

10.13.3 Commands for Feed Per Minute and Feed Per Revolution

In rigid tapping commands for feed per minute and feed per revolution can be specified.

(Example)
The example below specifies rigid tapping in feed per minute mode for cutting a thread with a lead of 1 mm at a spindle speed of 1,000 min^{-1}. (In feed per minute mode, F/S determines the thread lead.)

```
O0002 ;
G94 ;
  :
M29 S1000 ;
G84 X50. Y30. Z-100. R-20. F1000 ;
  :
G80 ;
```

The example below specifies rigid tapping in feed per rotation mode for cutting the same thread at the same spindle speed as above. (In feed per rotation mode, F determines the thread lead.)

```
O0001 ;
G95 ;
  :
M29 S1000 ;
G84 X50. Y30. Z-100. R-20. F1. ;
  :
G80 ;
```

NOTE
Even in the feed per revolution mode, pulses distributed for the drilling axis are converted to a command for feed per minute. Thus, feed per rotation mode does not strictly implement feed per rotation. Accordingly, even if the spindle stops for some reason, the drilling axis does not stop.

10.13.4 Acceleration/Deceleration after Interpolation

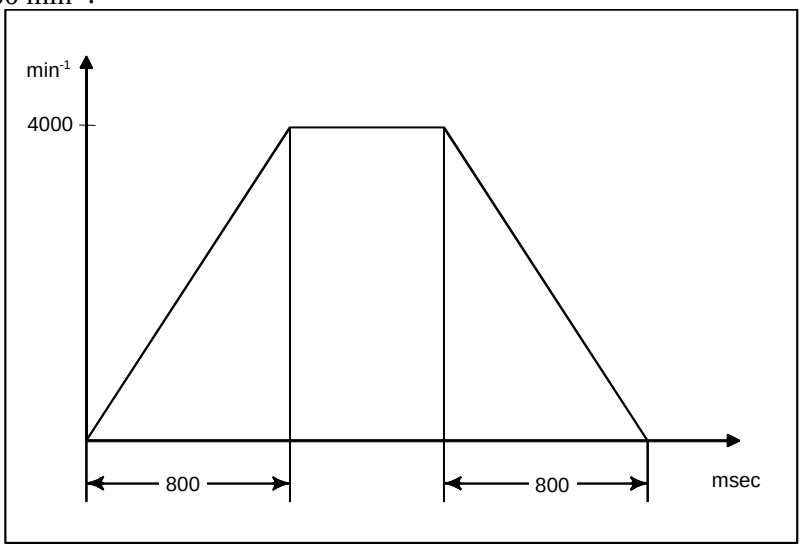
Linear acceleration/deceleration or bell-shaped acceleration/ deceleration can be applied.
When bit 2 (TDR) of parameter No. 5201 is set to 1, time constants for cutting and extraction can be set in separate parameters (Parameter No. 5271 to No. 5274).

NOTE
For using bell-shaped acceleration/deceleration, the option for rigid tapping bell-shaped acceleration/deceleration is required. The option for rigid tapping bell-shaped acceleration/deceleration is provided for the M series only.

- Linear acceleration/deceleration

By setting bit 5 (RBL) of parameter No. 5203 to 0, linear acceleration/deceleration of constant acceleration type can be applied.
In parameter No. 5261 to No. 5264, set a time required to reach a maximum spindle speed.
The actual time constant is the ratio between a maximum spindle speed and S command value.

Example)
Suppose that the parameters are set as follows: time constant TC = 800 msec and maximum spindle speed S = 4000 min^{-1}.



When S2000 is specified, the acceleration/deceleration time is 400 msec. When S1000 is specified, the acceleration/deceleration time is 200 msec. This means that the acceleration rate in linear acceleration/deceleration at rigid tapping time is constant.

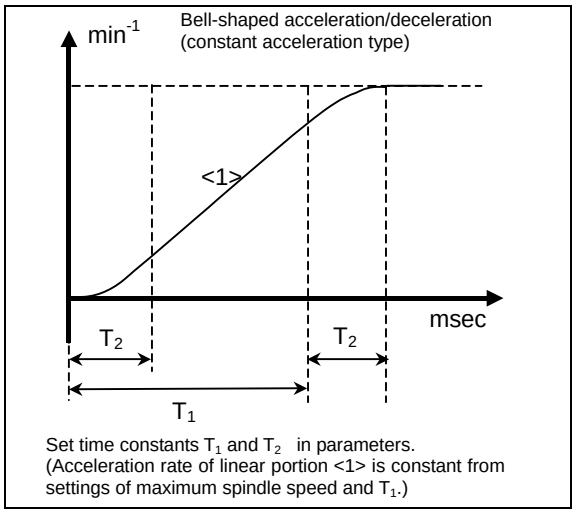
M

- Bell-shaped acceleration/deceleration

By setting bit 5 (RBL) of parameter No. 5203 to 1, bell-shaped acceleration/deceleration of constant acceleration type can be applied.

NOTE
For using bell-shaped acceleration/deceleration, the option for rigid tapping bell-shaped acceleration/deceleration is required.

In parameter No. 5261 to No. 5264, set a time required to reach a maximum spindle speed (linear portion).
In parameter No. 5365 to No. 5368, set a time for the curved portion.
The actual time constant for the linear portion is the ratio between a maximum spindle speed and S command value.
To the curved portion, not an S command value but a set value is applied.
This means that the actual time constant is [linear portion value obtained by proportional calculation + value for curved portion].



NOTE
1 Even when bit 2 (TDR) of parameter No. 5201 is set to 1, the values set in parameter No. 5365 to No.5367 are used for cutting and extraction in the curved portions.
2 When 0 is set in parameter No. 5365 to No.5367, linear acceleration/deceleration is used.

10.13.5 Override

Various types of overrides are disabled. However, the following overrides can be enabled by setting parameters:

- Extraction override
- Override signal

(1) Extraction override

For extraction override, a parameter-set fixed override value or programmed override value can be used at the time of extraction (including extraction operation at peck drilling/high-speed peck drilling).

- Parameter specification

Set bit 4 (DOV) of parameter No. 5200 to 1, and set an override value in parameter No. 5211.
An override value from 0% to 200% can be set in steps of 1%. When bit 3 (OVU) of parameter No. 5201 is set to 1, an override value from 0% to 2000% can be set in steps of 10%.

- Program specification

When bit 4 (DOV) of parameter No. 5200 and bit 4(OV3) of parameter No. 5201 are set to 1, a spindle speed to be used at extraction time can be programmed.
Using address J in a block specifying rigid tapping, specify a spindle speed to be used at extraction time.

Example)
When S = 1000 min^{-1} for cutting and S = 2000 min^{-1} for extraction
:
M29 S1000 ;
G84 Z-100. F1000. J2000 ;
:

The following calculation is made for conversion to an actual override value:

$$\text{Override value (\%)} = \frac{\text{Spindle speed for extraction } (J \text{ command})}{\text{Spindle speed } (S \text{ command})} \times 100$$

So, the spindle speed at extraction time may not match the speed specified by address J. If an override value not within the range 100% to 200% is specified, the specification of 100% is assumed.

By setting bit 6 (OVE) of parameter No. 5202 to 1, the override value can be expanded to the range from 100% to 2000%. In this case, if the override value exceeds the range 100% to 2000%, the override value is assumed to be 100%.

The table below summarizes the setting of an extraction override value by parameter specification/program specification.

When bit 6 (OVE) of parameter No. 5202 is set to 0

Command \ Parameter setting		DOV(No.5200#4)="1"		DOV (No.5200#4)=0
		OV3 (No.5201#4)=1	OV3 (No.5201#4)=0	
Spindle speed for extraction is specified by address J.	**Within the range 100% to 200%**	Program specification	Parameter No. 5211	100%
	Not within the range 100% to 200%	100%		
Spindle speed for extraction is not specified by address J.		Parameter No. 5211		

When bit 6 (OVE) of parameter No. 5202 is set to 1

Command \ Parameter setting		DOV(No.5200#4)="1"		DOV (No.5200#4)=0
		OV3 (No.5201#4)=1	OV3 (No.5201#4)=0	
Spindle speed for extraction is specified by address J.	**Within the range 100% to 2000%**	Program specification	Parameter No. 5211	100%
	Not within the range 100% to 2000%	100%		
Spindle speed for extraction is not specified by address J.		Parameter No. 5211		

NOTE

1 When specifying address J, do not use the decimal point.
If the decimal point is used, the specified value is handled as described below.
Example) When the increment system for the reference axis is IS-B
- When calculator-type decimal point input is not used
The specified value is converted to a value considering the least input increment.
"J200." is converted to 200000 min^{-1}.
- When calculator-type decimal point input is used
The specified value is converted to a value with the decimal places discarded.
"J200." is converted to 200 min^{-1}.

2 When specifying address J, do not use the minus sign.
When the minus sign is used, the allowable range is assumed to be exceeded.

NOTE

3 Find a maximum allowable extraction override value according to the expression below so that the spindle speed multiplied by an extraction override value does not exceed the maximum speed (parameter No. 5241 to No. 5244) of the gear in use. For this reason, a mismatch with the maximum spindle speed can occur, depending on the override value.

$$\text{Maximum override value (\%)} = \frac{\text{Maximum spindle speed (parameter setting)}}{\text{Spindle speed (}S\text{ command)}} \times 100$$

4 Address J for specifying a spindle speed for extraction, when specified in the rigid tapping mode, remains valid until the canned cycle is canceled.

(2) Override signals

When bit 4 (OVS) of parameter No. 5203 is set to 1, a cutting/extraction operation in rigid tapping can be overridden as described below.

- Override is applied using the feedrate override signal.
  (When the second feedrate override signal is valid, the second feedrate override is applied to the feedrate to which the feedrate override is already applied.
- Override is canceled using the override cancel signal.
  The relationship between this function and overriding each operation is described below.
- At the time of cutting
  When the override cancel signal = 0
  Value specified by the override signal
  When the override cancel signal = 1
  100%
- At the time of extraction
  When the override cancel signal = 0
  Value specified by the override signal
  When the override cancel signal = 1
  When extraction override is disabled
  100%
  When extraction override is enabled
  Value specified by an extraction override value

NOTE

Find a maximum allowable override value according to the expression below so that the spindle speed multiplied by an override value does not exceed the maximum speed (parameter No. 5241 to No. 5244) of the gear in use. For this reason, a mismatch with the maximum spindle speed can occur, depending on the override value.

$$\text{Maximum override value (\%)} = \frac{\text{Maximum spindle speed (parameter setting)}}{\text{Spindle speed (}S\text{ command)}} \times 100$$

10.13.6 Reference Position Return

For serial spindles, setting bit 0 (ORI) of parameter No. 5202 to "1" enables a reference position return to be made for a spindle at the start of rigid tapping.
When a movement is made to a R point in the rigid tapping mode, the reference position return direction is determined on the serial spindle side for reference position return. With the grid shift function, the reference position can be shifted within the range 0° to 360°.
Set a reference position return direction in bit 4 (RETSV) of parameter No. 4000. Set a grid shift amount in parameter No. 4073.

NOTE
1 A feedrate for reference position return is determined from the position where the tool is stopped when the rigid tapping mode is set, and the amount of travel found from the reference position. Accordingly, no fixed speed is applied.
2 When the gear ratio between the spindle and position coder is not $1:2^n$ (n: integer other than 0), the tool may not be able to return to the same reference position.

10.13.7 FS10/11 Format Command

M

When bit 1 (FCV) of parameter No. 0001 is set to 1, rigid tapping can be specified in the FS10/11 command format.

```
1)   :
     Sxxxx ;
     G84.2/G84.3 X_ Y_ Z_ R_ P_ F_ L_ ;
      :
2)   :
     G84.2/G84.3 X_ Y_ Z_ R_ P_ F_ L_ Sxxxx ;
      :
```

G84.2: Rigid tapping cycle
G84.3: Rigid reverse tapping cycle
X,Y : Tapping position
Z : Position of the bottom face of a tapped hole
R : Position of a R point
P : Dwell time at the bottom of a tapped hole and at R point return time
F : Cutting feedrate
L : Number of repeats
S : Spindle speed

NOTE
This function enables the rigid tapping function to be specified in the FS10/11 command format without changing the rigid tapping sequence (such as handling with the PMC).

T

When bit 1 (FCV) of parameter No. 0001 is set to 1 and bit 3 (F0C) of parameter No. 5102 is set to 0, rigid tapping can be specified in the Series 10/11 command format.

```
1)   :
     Sxxxx ;
     G84.2  X_  C_  Z_  R_  P_  F_  L_  ;
     :
2)   :
     G84.2  X_  C_  Z_  R_  P_  F_  L_  S_;
     :
```

G84.2: Rigid tapping cycle G code
X,C : Tapping position
Z : Position of the bottom face of a tapped hole

R : Position of a R point
P : Dwell time at the bottom of a tapped hole and at R point return time
F : Cutting feedrate
L : Number of repeats
S : Spindle speed

The differences from the FS9 command format are as follows:

- No G code can make a distinction between a front tapping cycle and side tapping cycle. A tapping axis is determined by plane selection (G17/G18/G19). Select a plane that matches a front tapping cycle/side tapping cycle.

Plane select	Drilling axis
G17 Xp-Yp plane	Zp
G18 Zp-Xp plane	Yp
G19 Yp-Zp plane	Xp

Xp : X axis or its parallel axis
Yp : Y axis or its parallel axis
Zp : Z axis or its parallel axis

NOTE
With bit 0 (FXY) of parameter No. 5101, the Z-axis can be used as the tapping axis at all times. If FXY is set to 0, the Z-axis is used as the tapping axis at all times.

- The M code for C-axis clamping cannot be specified.
- The specification of a R point depends on the G code system and parameter setting as described below.

1. When the parameter F0C (No.5102#3) is set 0.

Parameter RAB (No.5102#6)		
0	**1**	
Incremental command at all times	**G code system**	
	A	**B,C**
	Absolute command at all times	Follows G90/G91.

Parameter RDI(No.5102#7)	
0	**1**
Diameter command at all times	Follows the diameter specification/radius specification of the tapping axis.

2. When the parameter F0C (No.5102#3) is set 1.
An incremental command based on radius specification is used, regardless of bit 6 (RAB) and bit 7 (RDI) of parameter No. 5102.

NOTE
This function enables the rigid tapping function to be specified in the Series 10/11 command format without changing the rigid tapping sequence (such as handling with the PMC).

10.13.8 Multi Spindle Control

When multi spindle control is enabled, rigid tapping can be performed by selecting spindles from the first to second spindles. A spindle can be selected as described below by parameter setting.

NOTE
To enable multi-spindle control, set bit 3 (MSP) of parameter No. 8133 to 1.

		T series	
		Parameter SRS(No.5200#7)	
		0	1
Parameter MPP (No.3703#3)	0	Selected using the spindle selection signals SWS1 to SWS2	Selected using the rigid tapping spindle selection signals RGTSP1 to RGTSP2
	1	Selected using an NC command (address P)	

For details of the method of selection using the spindle selection signals/rigid tapping spindle selection signals, see the descriptions of signals provided later.
For details of the method of selection using an NC command (address P), see Section 10.12 "MULTI-SPINDLE CONTROL".

10.13.9 Rigid Tapping with Spindle of Another Path (T Series (2-ptah Control))

T

Overview

In 2-path system, rigid tapping with spindle of other path different from specified path can be executed by path spindle control function. The spindle which executes rigid tapping is selected according to the spindle commandselection signals.
If multi-spindle control option is provided, not only the path but also the spindle executed with rigid tapping can be selected.

Specifications

- Case in which no multi-spindle function is used

If bit 4 (SPR) of parameter No. 3703 is set to "1", using spindle command select signals, like the S command, makes it possible to specify a path from which rigid tapping command is to be used to control a spindle in a specific path.
The spindle command select signals are described below.
(Example)

2-path system: SLSPA<G063.2> (for a spindle in path 1) and SLSPB<G063.3> (for a spindle in path 2)

- Case in which the multi-spindle function is used

- Signal-based command

If bit 1 (EMS) of parameter No. 3702 is set to "0" and bit 4 (SPR) of parameter No.3703 is set to "1", enables an arbitrary spindle in an arbitrary path to be used for rigid tapping.
Before a rigid tapping command is issued, the spindle to be used for rigid tapping is selected using a signal.

- Address P-based command

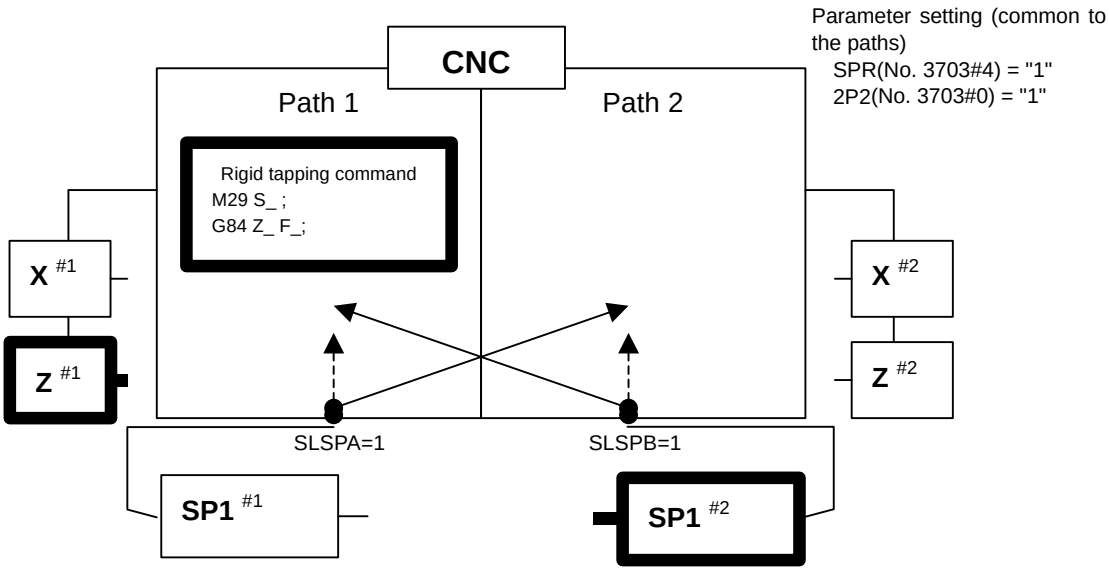
  If bit 1 (EMS) of parameter No. 3702 is set to "0" and bit 4 (SPR) of parameter No.3703 is set to "1", makes it possible to use, for rigid tapping, a spindle connected to a path other than the one in which the command is issued. Setting bit 3 (MPP) of parameter No. 3703 to "1" enables a spindle to be selected for use in rigid tapping according to a program (P code).
  Adding address P to the S command for rigid tapping enables a spindle to be selected for use in rigid tapping.
  (Example: **M29 S1000 P22 ;**)
  It is necessary to set, in advance, parameter No. 3781 with relationships between P values and axes selected with specific P values.

  Refer to descriptions of Section, "MULTI-SPINDLE CONTROL" in this manual for explanations about how to use address P to select a spindle.

Example 1 Case in which no multi-spindle control is used

On a 2-path lathe with 2 spindles under 2-path control (bit 0 (2P2) of parameter No. 3703 is set to "1"), the following signals are set up as stated when rigid tapping is carried out using the rigid tapping command (machining program) and servo section (Z $^{\#1}$ in the following diagram) in path 1 and the first spindle (SP1 $^{\#2}$ in the following diagram) connected to path 2.

<1> SLSPB<G63.3> =1: The spindle connected to path 2 is controlled by commands in path 1.
<2> SLSPA<G63.2> = 1: The spindle connected to path 1 is controlled by commands in path 2.
Rigid tapping is carried out on the first spindle (SP1$^{\#2}$) in path 2.



(Supplement)

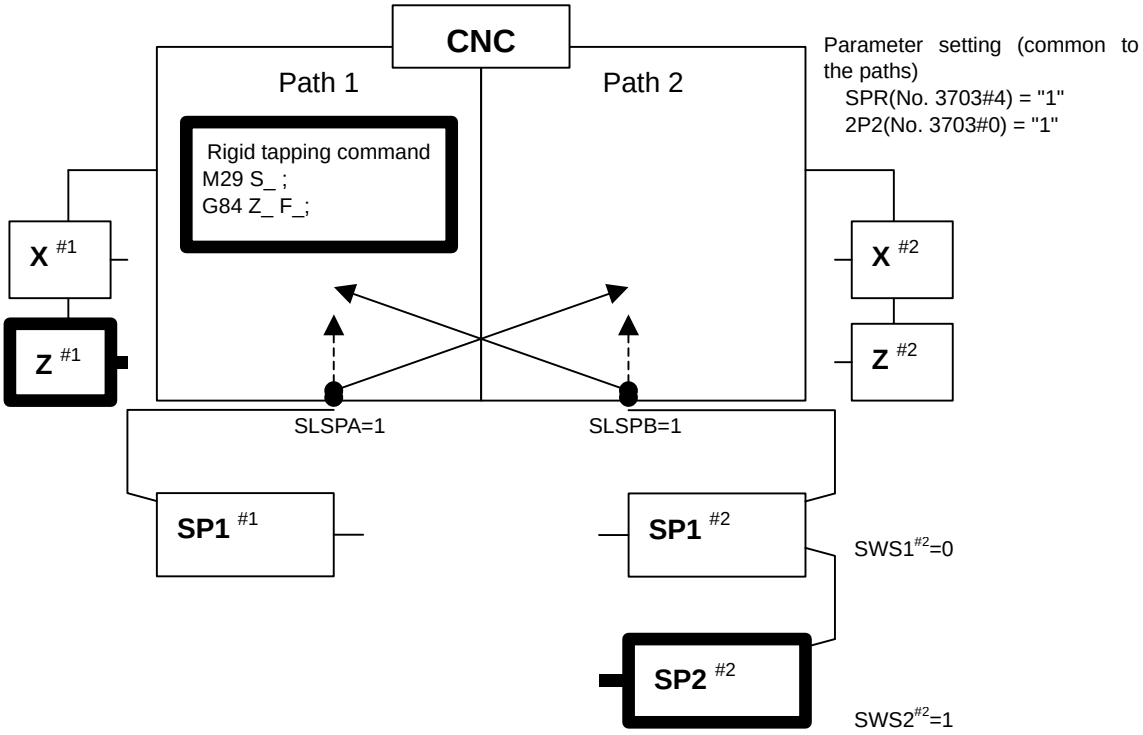
In the above example system configuration, the signal operation described in <2> is necessary to make each command correspond to the spindles selected by the commands on a one-to-one basis.
The way the spindle select signals are handled may vary according to the number of control paths in the CNC and the number of controlled axes (bits 0 (2P2) and 2 (MPM) of parameter No. 3703).

Example 2 Case in which multi-spindle control is used (signal-based command)

On a 2-path lathe with 3 spindles under 2-path control (bit 0 (2P2) of parameter No. 3703 is set to "1"), the following signals are set up as stated when rigid tapping is carried out using the rigid tapping command (machining program) and servo section (Z $^{\#1}$ in the following diagram) in path 1 and the second spindle (SP1 $^{\#2}$ in the following diagram) connected to path 2.

<1> SLSPB<G63.3> = 1: The spindle connected to path 2 is controlled by commands in path 1.
<2> SLSPA<G63.2> = 1: The spindle connected to path 1 is controlled by commands in path 2.
<3> SWS1#2<G1027.0> ＝0

<4> SWS2#2<G1027.1> =1: Rigid tapping is carried out on the second spindle (SP2#2) in path 2.



(Supplement)

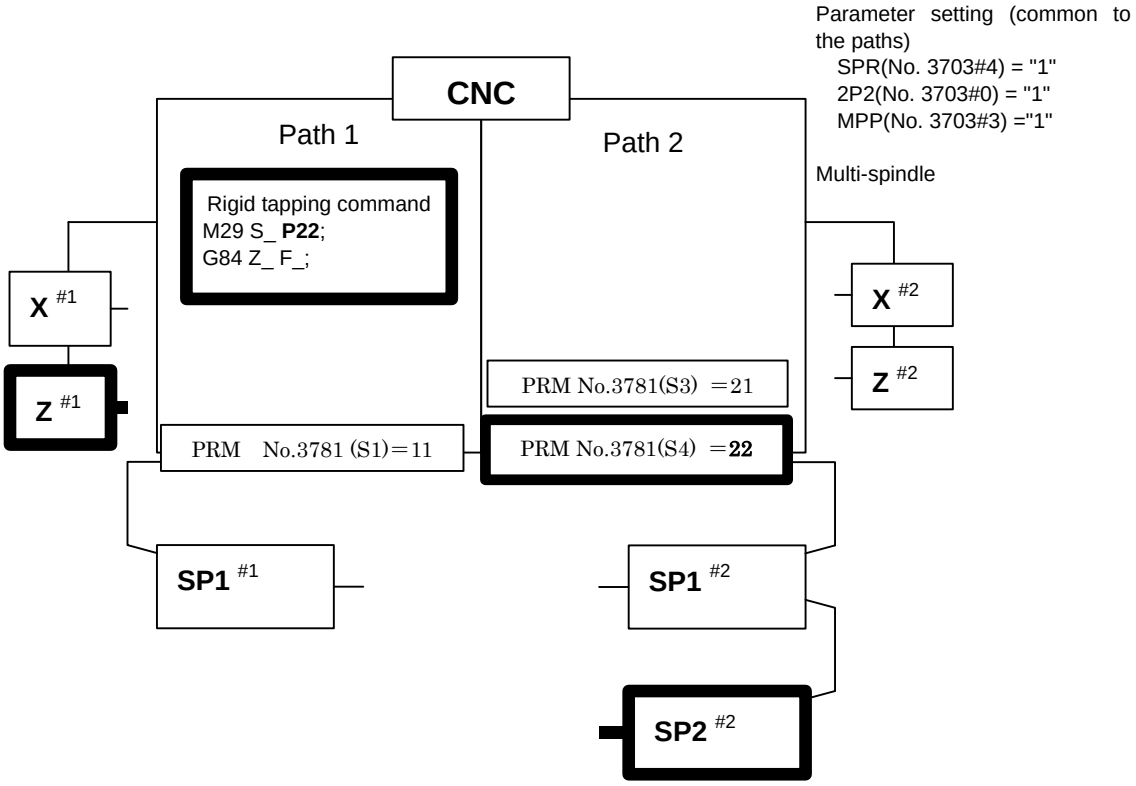
In the above example system configuration, the signal operation described in <2> is necessary to make each command correspond to the spindles selected by the commands on a one-to-one basis.
The way the spindle select signals are handled may vary according to the number of control paths in the CNC and the number of controlled axes (bits 0 (2P2) and 2 (MPM) of parameter No. 3703).

Example 3

Case in which multi-spindle control is used (address P-based command)

On a 2-path lathe with 3 spindles under 2-path control (bit 0 (2P2) of parameter No. 3703 is set to "1"), before a rigid tapping command is issued, the following parameters are set up as stated if rigid tapping is to be carried out using the rigid tapping command (machining program) and servo section (Z #1 in the following diagram) in path 1 and the second spindle (SP2 #2 in the following diagram) connected to path 2.

No.3781#1 S1 (Path 1: P value for selecting the first spindle) = 11
No.3781#2 S2 (Path 2: P value for selecting the first spindle) = 21
No.3781#2 S3 (Path 2: P value for selecting the second spindle) = 22



The P code for selecting a spindle is added to the rigid tapping S command to execute a machining program. In this example, SP2 #2 must be controlled. So, the P code is used to issue a value (22) specified by No. 3781#2 (S4).

Restrictions

Observe the following restrictions.

- Paths issuing rigid tapping commands must correspond to spindles on which rigid tapping is carried out on a one-to-one basis.
- If this function is used to carry out rigid tapping on a spindle in a path other than a command-issuing path, the spindle must be put in the spindle control mode in advance.
- This function is unusable for servo motor-based rigid tapping.

The alarms below are raised if a restriction above is not observed, that is, if a condition stated below occurs.

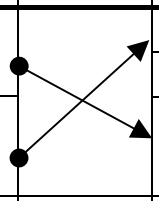
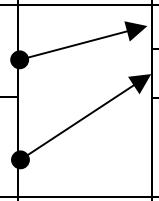
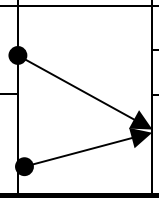
Alarm PS0205 "RIGID MODE DI SIGNAL OFF"

- Rigid tapping commands in multiple paths attempted to carry out rigid tapping on the same spindle simultaneously.
- A rigid tapping command was issued to a spindle control axis with servo motor in a path other than the command issuing path.

Alarm SP0752 "SPINDLE MODE CHANGE ERROR"

- A spindle selected by a rigid tapping command was in a mode other than the spindle control mode.
  The modes other than the spindle control mode include:
  Cs contour axis
  Spindle positioning
  Spindle-synchronous, inter-spindle polygon machining
  Rigid tapping (controlled by commands from another path)

[Example] When a 2-path machine has two spindles in path 1 and one spindle in path 2 (Rigid tapping is carried out on a spindle pointed to by an arrowhead)

	Path issuing commands		Spindle on which rigid tapping is carried out	MPP = 0	MPP = 1
Example 1	Path 1 Path 2		First spindle Second spindle First spindle	Available	Available
Example 2	Path 1 Path 2		First spindle Second spindle First spindle	Unavailable Alarm PS0205	Available
Example 3	Path 1 Path 2		First spindle Second spindle First spindle	Unavailable Alarm PS0205	Unavailable Alarm PS0205

If the rigid tapping spindle is not in the spindle control mode (for example, is in the Cs contour control mode), rigid tapping cannot be carried out in any of the above cases.

10.13.10 Optimum Acceleration/Deceleration for Rigid Tapping

Overview

With this function, the acceleration/deceleration applied during cutting in rigid tapping can be set flexibly according to the torque characteristic of the spindle motor and machine characteristics such as friction in the machine.

Normally, because of the torque characteristic of the spindle motor and machine characteristics, the low-speed portion and high-speed portion of acceleration/deceleration performance that can be output (referred to as the maximum acceleration curve hereinafter) are not symmetric. Since conventional acceleration/deceleration (linear and bell-shaped types) was symmetric, it was impossible to maximize the motor capability. This function allows acceleration/deceleration to be applied so that the actual acceleration/deceleration curve can follow the maximum acceleration/deceleration curve as closely as possible. This can maximize the motor capability and reduce cutting time. When rigid tapping is performed when the acceleration rate of the spindle motor is kept constant, however, reduction in cutting time cannot be expected.

When acceleration patterns are set in parameters for individual gears, rigid tapping can be performed with acceleration/deceleration applied according to the maximum acceleration curve.

At the time of extraction, the acceleration pattern used for cutting is used to apply acceleration/deceleration.

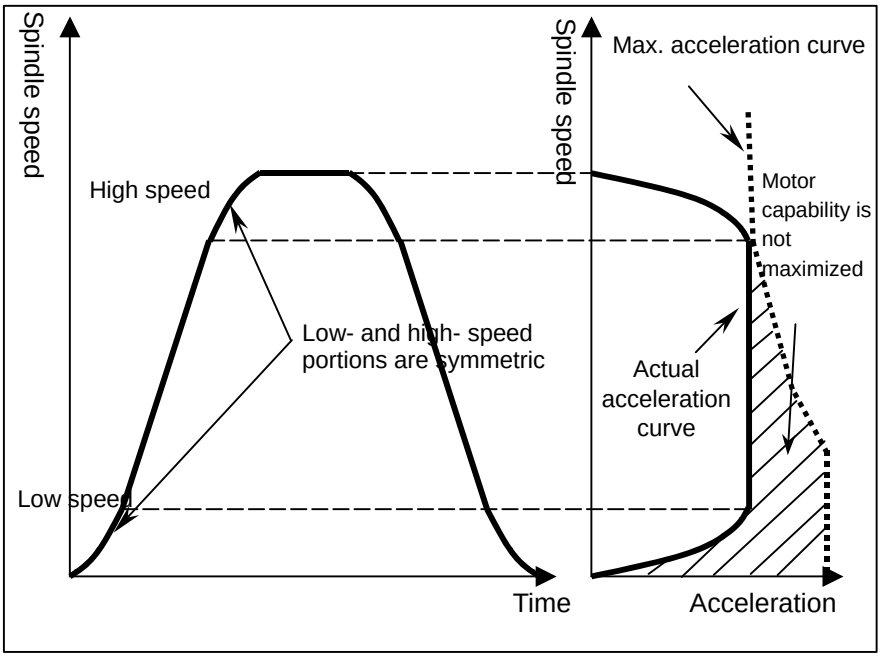


Fig. 10.13.10 (a) Conventional acceleration/deceleration (bell-shaped)

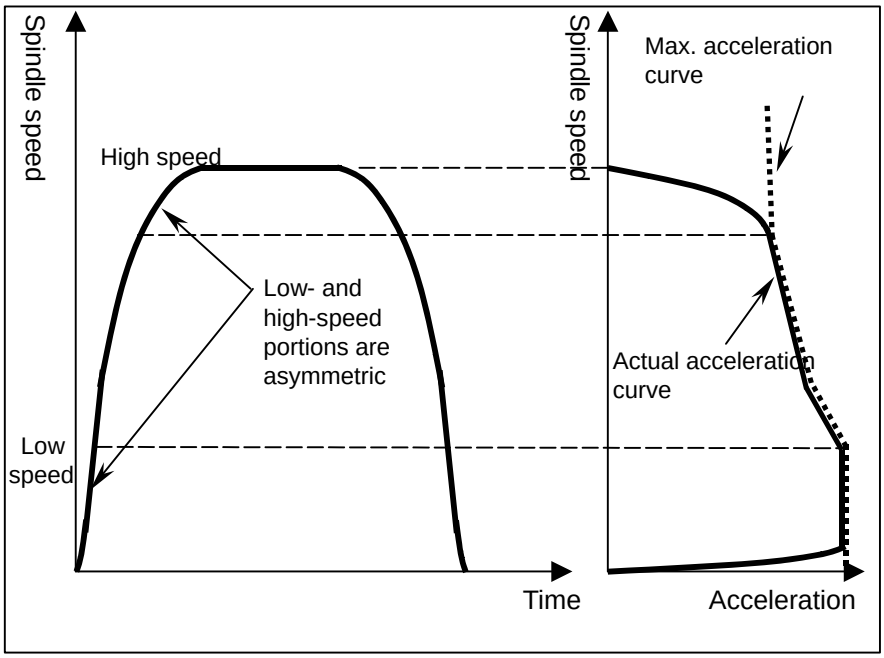


Fig. 10.13.10 (b) Acceleration/deceleration controlled by this function, in which the actual acceleration curve follows the maximum acceleration curve

NOTE
The rigid tapping optimum acceleration/deceleration function is an optional function.

Description

Based on the spindle acceleration/deceleration status and the selected gear (between the spindle and spindle motor), this function selects a parameter-set acceleration/deceleration pattern, obtains a permissible acceleration rate of the spindle from the current spindle speed, and controls acceleration/deceleration not to exceed the permissible acceleration rate of the spindle, to perform rigid tapping.

NOTE
In this subsection, up to four gear stages are assumed between the spindle and spindle motor in rigid tapping. The T series can use up to four gear stages, but the M series can use up to only three gear stages.

- Setting optimum acceleration/deceleration

Set bit 0 (RAU) of parameter No. 11420 to 1 to enable this function.

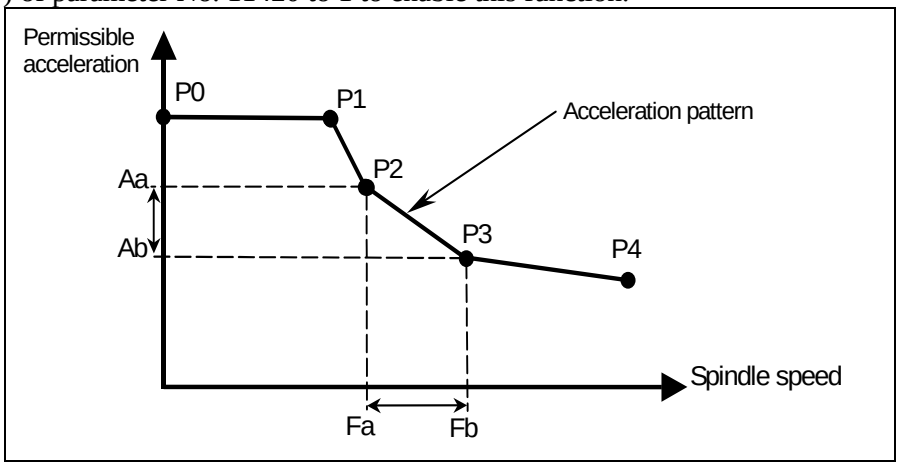


Fig. 10.13.10 (c) Acceleration setting points

In parameters provided for each gear, set a speed and a permissible acceleration rate at each acceleration setting point (P0 to P4) for both spindle acceleration and deceleration.
The line connecting the acceleration setting points is an acceleration pattern.
The permissible acceleration changes, for example, from Aa to Ab according to the spindle speed while the speed changes from Fa to Fb in the above figure.

Table 10.13.10 (a) Gear-by-gear acceleration pattern parameters

Gear	Max. acceleration parameter No.	Max. spindle speed parameter No.	Acceleration setting point	Spindle speed parameter No.	Permissible acceleration parameter No.	
					Forward/reverse rotation	
					During acceleration	During deceleration
Gear 1	11421	5241	P0	None (0%)	11441	11461
			P1	11429	11442	11462
			P2	11430	11443	11463
			P3	11431	11444	11464
			P4	None (100%)	11445	11465
Gear 2	11422	5242	P0	None (0%)	11446	11466
			P1	11432	11447	11467
			P2	11433	11448	11468
			P3	11434	11449	11469
			P4	None (100%)	11450	11470
Gear 3	11423	5243	P0	None (0%)	11451	11471
			P1	11435	11452	11472
			P2	11436	11453	11473
			P3	11437	11454	11474
			P4	None (100%)	11455	11475
Gear 4	11424	5244	P0	None (0%)	11456	11476
			P1	11438	11457	11477
			P2	11439	11458	11478
			P3	11440	11459	11479
			P4	None (100%)	11460	11480

- Set parameters for the gear used.
- Set the maximum acceleration rate (rev/sec^2) in a parameter (No. 11421 to No.11424). When 0 is set in this parameter, alarm DS1711 is issued at the time of execution.
- At P0, the spindle speed need not be set because 0% is assumed.
- At P4, the spindle speed need not be set because 100% is assumed.
- Set the spindle speeds at P1 to P3 in parameter Nos. 11429 to 11440 with a percentage to the maximum spindle speed (set in parameter No. 5241 to 5244) applied during rigid tapping.
- Set the permissible acceleration rates at P0 to P4 in parameter Nos. 11441 to 11480 with a percentage to the maximum acceleration rates(set in parameter Nos. 11421 to 11424).
- When the spindle speed is set to 0 in parameter No. 11429 to 11440 for an acceleration setting point, that point is skipped, and the acceleration pattern line is connected to the next acceleration setting point that has a non-zero value.
- When the permissible acceleration rate is set to 0 in parameter No. 11441 to 11480, it is assumed to be 100%.

- Formula indicating the relationship between the torque and acceleration

If the torque is $T(N \cdot m)$, and the inertia is $J(kg \cdot m^2)$, acceleration rate $A(rev/\sec^2)$ is calculated as follows:

$$\begin{aligned} A &= T\,(N \cdot m)\,/\,J\,(kg \cdot m^2)\,/\,(2\pi) \\ &= T\,((kg \cdot m/\sec^2)(m))/\,J\,(kg \cdot m^2)/(2\pi) \\ &= T/((2\pi)\cdot J) \end{aligned}$$

- Example for setting an acceleration pattern

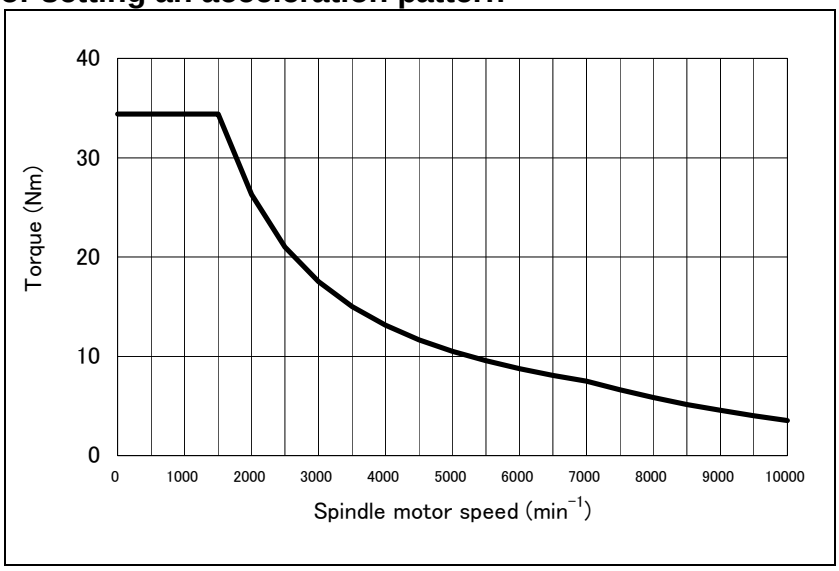


Fig. 10.13.10 (d) Speed-torque characteristic in the 30-min, S3 60% operation area of model αiI3/10000

Structure

- The spindle and spindle motor are directly coupled.
- The spindle inertia is almost equal to the rotor inertia.
- The total inertia is twice as large as the rotor inertia.
- Model αiI3/10000 motor

Model αiI3/10000 motor specifications	
Rotor inertia	0.0148$(kg \cdot m^2)$
Maximum torque	34.3$(N \cdot m)$; Speed 0 to 1500 $(\min^{-1})$
Spindle inertia	0.0148$(kg \cdot m^2)$

From the above formula indicating the torque-acceleration relationship, acceleration rate $A\left(rev/\sec^2\right)$ to be applied when the maximum torque $34.3\left(N \cdot m\right)$ is observed is obtained.

$$A = 34.3/((2\pi)\cdot 2.0 \cdot 0.0148)\left(rev/\sec^2\right) = 184.426\left(rev/\sec^2\right)$$

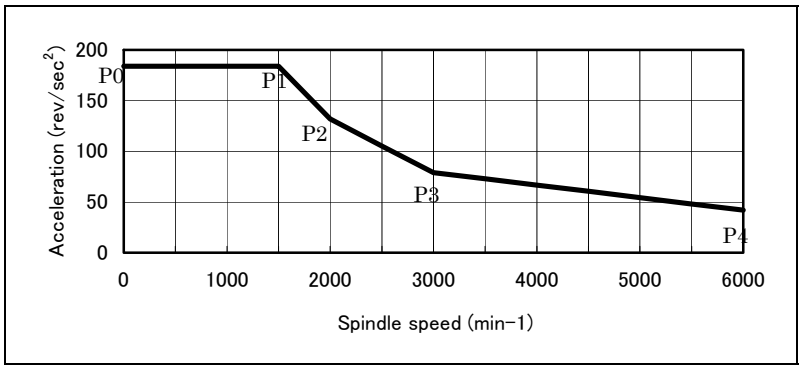
An example of setting parameters for an acceleration pattern is provided in the table below.
Setting conditions

- The same acceleration rate is used for acceleration and deceleration.
- The spindle is directly coupled with the spindle motor (1:1).
- The maximum spindle speed is 6000 min^{-1}.

Table 10.13.10 (b) Example of setting parameters for an acceleration pattern

	Parameter No.	Setting (unit)	Remark
Maximum spindle speed	5241	6000 (min^{-1})	—
Maximum acceleration	11421	184 (rev/sec^2)	Maximum acceleration A is set.
Spindle speed (P1)	11429	25 (%)	Because torque 34.3 (Nm) retains when the speed is 1500 (min^{-1}) or less, 25% (1500min^{-1}) is set for P1.
Spindle speed (P2)	11430	33 (%)	Because the torque drops almost linearly when the speed changes from 1500 (min^{-1}) to 2000 (min^{-1}), 33% (1980min^{-1}) is set for P2.
Spindle speed (P3)	11431	50 (%)	Because the torque drops almost linearly when the speed changes from 2000 (min^{-1}) to 3000 (min^{-1}), 50% (3000min^{-1}) is set for P3.
Acceleration (P0)	11441 11461	100 (%)	Acceleration applied to speed 0 is set to 100% (with a torque of 34.3 (Nm)).
Acceleration (P1)	11442 11462	100 (%)	Acceleration applied to speed at P1 is set to 100% (with a torque of 34.3 (Nm)).
Acceleration (P2)	11443 11463	72 (%)	Acceleration applied to speed at P2 is set to 72% (with a torque of 25 (Nm))..
Acceleration (P3)	11444 11464	43 (%)	Acceleration applied to speed at P3 is set to 43% (with a torque of 15 (Nm)).
Acceleration (P4)	11445 11465	23 (%)	Acceleration applied to speed at P4 is set to 23% (with a torque of 8 (Nm)).

When the above parameter settings are made, the obtained acceleration pattern is plotted as follows:



At P0, the spindle speed is 0 min^{-1}, and the acceleration rate is 184 rev/sec^2.
At P1, the spindle speed is 1500 min^{-1}, and the acceleration rate is 184 rev/sec^2.
At P2, the spindle speed is 1980 min^{-1}, and the acceleration rate is 132 rev/sec^2.
At P3, the spindle speed is 3000 min^{-1}, and the acceleration rate is 79 rev/sec^2.

At P4, the spindle speed is 6000 min^{-1}, and the acceleration rate is 42 rev/sec^2.

NOTE
1 An acceleration pattern must be set according to the spindle motor specifications, machine characteristics, and so on.
2 Maximum spindle speed parameter Nos. 5241 to 5244 and spindle speed parameter Nos. 11429 to 11440 do not sometimes match the actual motor speed because of the gear between the spindle and spindle motor. Therefore, an acceleration pattern must be set considering the actual spindle motor speed. For example, when the spindle motor speed is one half of the spindle speed, the acceleration rate applied when the spindle speed is 6000 min^{-1} is determined by the torque observed when the motor speed is 3000 min^{-1}.
3 Before modifying an acceleration pattern, ensure that rigid tapping optimum acceleration/deceleration is not being performed in either path.

- Bell-shaped acceleration/deceleration

When bell-shaped acceleration/deceleration is to be applied, set bell-shaped acceleration change time in parameter No. 11425 to 11428.
When bell-shaped acceleration/deceleration is not to be performed, set 0.

- Acceleration/deceleration after interpolation

When rigid tapping optimum acceleration/deceleration is enabled, acceleration/deceleration after interpolation is not applied.

- Override

Various types of overrides are disabled. However, the following overrides can be enabled by parameter setting:

- Extraction override
- Override signal

For details, refer to "Override" in a previous subsection.

- Retraction for rigid tapping

This function is effective in case of the retraction for rigid tapping.

- Functions that can be used at the same time

Optimum acceleration/deceleration for rigid tapping can be used with the following functions at the same time:

- Axis synchronous control
- Tandem control
- Composite control (T series)
- Multi-spindle control
- Simple spindle synchronous control (M series)

- Maximum cutting feedrate

The maximum cutting feedrate parameter of the acceleration / deceleration before the interpolation (No. 1432) is effective in this function. Set the parameter of the drilling axis.

Restrictions

- Rigid tapping type

Optimum acceleration/deceleration for rigid tapping cannot be used with the following rigid tapping functions:

- Rigid tapping with an analog spindle

- Rigid tapping with a servo motor

- Functions that cannot be used at the same time

Optimum acceleration/deceleration for rigid tapping cannot be used with the following functions at the same time:

- General purpose retract
  Retract signal RTRCT is ignored while rigid tapping optimum acceleration/deceleration is being performed.

- Acceleration pattern

- An acceleration pattern can be set for acceleration and deceleration, but the ratio of the deceleration rate to the acceleration rate must be at least 1/3. For example, when acceleration at an acceleration setting point (Pn) is set to 90%, the deceleration rate at that point must be set to 30% or more.
- An acceleration pattern for deceleration must be set so that time it takes to decelerate the spindle speed from the maximum spindle speed to 0 is 4000 msec or less. However, this time does not include time required for changing the acceleration rate in bell-shaped acceleration/deceleration.
- If the ratio of the deceleration rate to the acceleration rate or deceleration time until the speed becomes 0 is set to a value beyond the permissible range, alarm DS1711 is issued at the time of execution.
- There is some calculation difference between the set acceleration rate and the actual acceleration rate.

10.13.11 Notes

- This function is enabled when a serial spindle is used. When any spindle in another path is to be specified, multi-spindle control must be enabled.
- If rigid tapping is carried out on a spindle in a path other than a command issuing path, the ENB signal is also shifted to the path to which the spindle is connected. Be careful when using the spindle ENB signal in the rigid tapping release sequence.
- Be careful not to issue a spindle mode switching command from one path and a rigid tapping command from another path to the same spindle simultaneously (for example, using an inter-path wait M code).

Example: The Cs contour control function is executed on the first spindle in path 1. After that, a rigid tapping command is issued from path 2 to that spindle.

Commands in path 1		Commands in path 2
: M20 ; (Enter Cs contour control mode) G00 C90.0 ; (C-axis positioning) M21 ; (Exit from Cs contour control mode) M12000 ; : :	← Waiting →	: : : : M12000 ; M29 S_ P11 ; (Rigid tapping) G84 X_ Z_ F_ ;

M12000: Inter-path wait M code (see descriptions of parameter Nos. 8110 and 8111)
Parameter No.3781#1 = 11

- If this function is used to carry out rigid tapping on a spindle connected to a path other than a command issuing path, the settings in the path to which the spindle is connected are applied to the following parameters.
  No.5214 (synchronization error width)
  Nos.5221 to 5224 (number of gear teeth on spindle side)
  Nos.5231 to 5234 (number of gear teeth on PC side)
  Nos.5241 to 5244 (maximum rotation speed)

Nos.5261 to 5264 (acceleration/deceleration constant)
Nos.5271 to 5274 (acceleration/deceleration constant for retracting)
Nos.5280 to 5284 (loop gain)
No.5300, No.5302 (spindle effective area (for in-position check))
No.5310, No.5350 (excessive error during spindle movement)
No.5312, No.5352 (excessive error with spindle at stop)
Nos.5321 to 5324 (backlash)

10.13.12 Diagnosis Display

For rigid tapping adjustment, the diagnosis screen displays information related to rigid tapping.

- Display of command pulses and position deviation amounts

- Spindle position deviation → Diagnosis display No.450
- Number of command pulses distributed to the spindle (momentary value) → Diagnosis display No.451
- Cumulative number of command pulses distributed to the spindle → Diagnosis display No.454

- Display of rigid tapping synchronization error

- Spindle-converted move command difference → Diagnosis display No.455
- Spindle-converted position deviation difference → Diagnosis display No.456
- Synchronization error range → Diagnosis No.457
- Spindle-converted move amount difference (maximum value) → Diagnosis display No.460
- Spindle-converted machine position difference (momentary value) → Diagnosis display No.461
- Spindle-converted machine position difference (maximum value) → Diagnosis display No.462

$$\text{Spindle - converted move command difference} = \Sigma \frac{\text{spindle move command}}{\text{gear ratio}} - \Sigma \frac{(\text{tapping axis move command}) \times \text{number of pulses per spindle revolution}}{\text{thread lead}}$$

$$\text{Spindle - converted position deviation difference} = \frac{\text{spindle position deviation}}{\text{gear ratio}} - \frac{(\text{drilling axis position deviation}) \times \text{number of pulses per spindle revolution}}{\text{thread lead}}$$

Synchronization error range =
(maximum spindle - converted move position deviation difference on the positive side) -
(maximum spindle - converted position deviation difference on the negative side)

$$\text{Drilling axis machine position} = \Sigma [\text{drilling axis move command}] - \text{drilling axis position deviation}$$

$$\text{Spindle machine position} = \Sigma [\text{spindle move command}] - \text{spindle position deviation}$$

$$\text{Spindle - converted machine position deviation difference} = \frac{\text{drilling axis machine position} \times \text{number of pulses per spindle revolution}}{\text{thread lead}} - \frac{\text{spindle machine position}}{\text{gear ratio}}$$

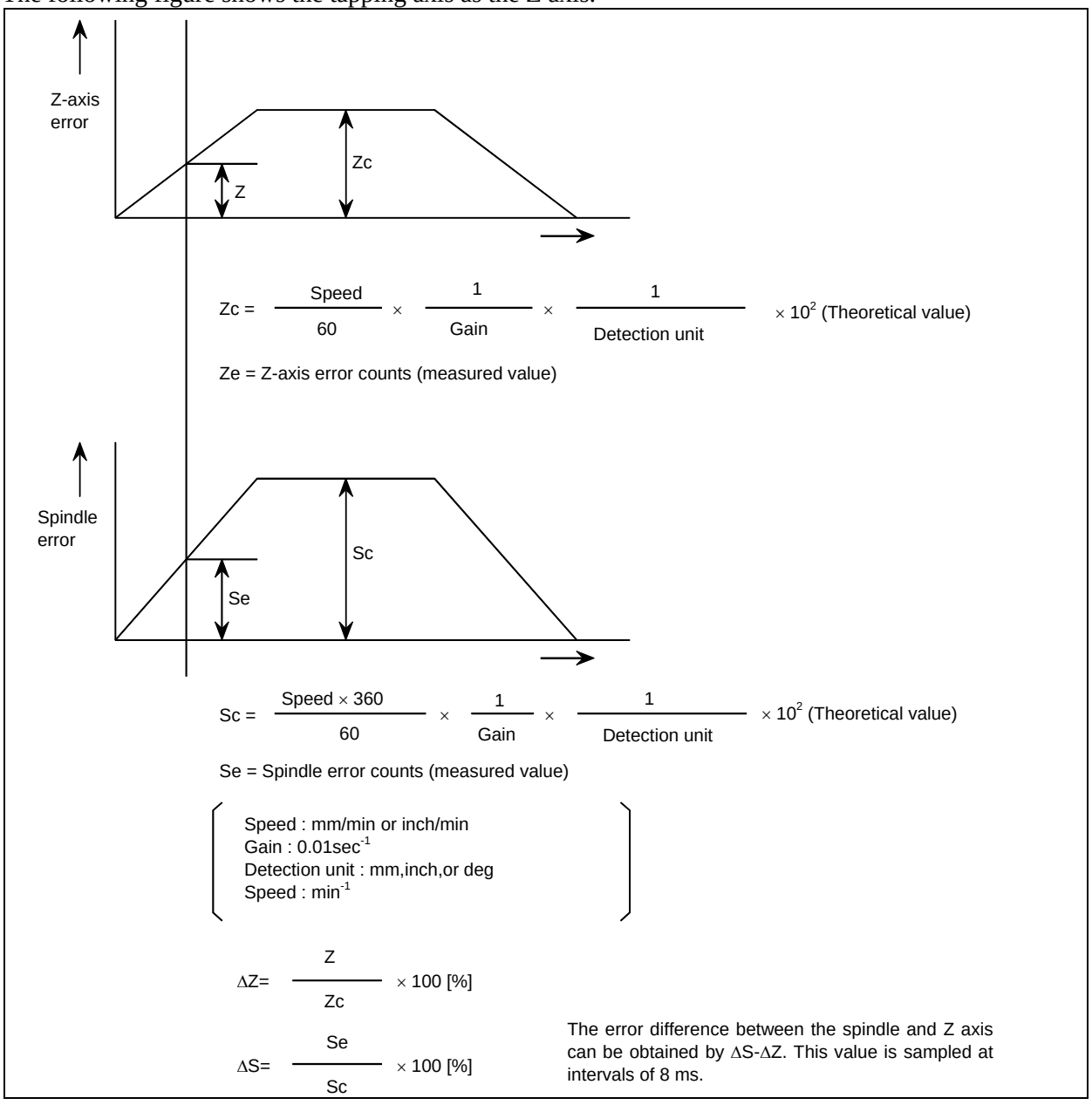
If a maximum allowable synchronization error range is set in parameter No. 5214, the position deviation alarm during spindle movement (alarm SP0741) is issued to indicate that the set synchronization error range has been exceeded. (If 0 is set in parameter No. 5214, no check is performed to detect whether the synchronization error range has been exceeded.)

- Error difference display

- Momentary error difference between the spindle and drilling axis → Diagnosis display No.452
- Maximum error difference between the spindle and drilling axis → Diagnosis display No.453

Diagnosis display No. 0452 is cleared to 0 at the beginning and end of the rigid tapping mode. Diagnosis display No. 0453 is cleared to 0 by the positioning cycle of the rigid tapping cycle.

The following figure shows the tapping axis as the Z axis.



- Display of spindle position data

Position coder signal pulse data from the spindle one-rotation signal is displayed.

- Position coder signal pulse data of each spindle → Diagnosis display No.445

NOTE
1 For an unconnected spindle, 0 is displayed.
2 For display of this data, the conditions below must be satisfied.
<1>Serial spindle
<2>Bit 1 (SPP) of parameter No. 3117 must be set to 1.
<3>Use a spindle amplifier and serial spindle manufactured by FANUC.
<4>The state where the serial spindle has detected a one-rotation signal must be set.
To detect a one-rotation signal accurately, spindle orientation must be performed. This orientation operation needs to be performed only once after the power is turned on.
Whether a one-rotation signal has been detected can be known by checking the serial spindle state signals (PC1DEA to PC1DEB).

Diagnosis display

- Spindle position data

0445	Spindle position data

[Data type] Word spindle
[Unit of data] Pulse
When a serial spindle is used, position coder signal pulse data from the one-rotation signal is indicated as the position data of the spindle.
This parameter is valid when bit 1 (SPP) of parameter No. 3117 is set to 1.
Before spindle position data can be displayed, spindle orientation must be performed once.

- Spindle position deviation

0450	Position deviation of the spindle during rigid tapping

[Data type] 2-word spindle
[Unit of data] Detection unit

- Number of pulses distributed to the spindle

0451	Number of pulses distributed to the spindle during rigid tapping

[Data type] 2-word spindle
[Unit of data] Detection unit

- Error difference between the spindle and drilling axis (momentary value)

0452	Difference in error between the spindle and drilling axis during rigid tapping (momentary value)

[Data type] 2-word spindle
[Unit of data] %

- Error difference between the spindle and drilling axis (maximum)

0453	Difference in error between the spindle and drilling axis during rigid tapping (maximum value)

[Data type] 2-word spindle
[Unit of data] %

- Cumulative number of pulses distributed to the spindle during drilling tapping

0454	Number of pulses distributed to the spindle during rigid tapping (cumulative number)

[Data type] 2-word spindle
[Unit of data] Detection unit

- **Spindle-converted move command difference during rigid tapping (momentary value)**

0455	Spindle-converted move command difference during rigid tapping (momentary value)

[Data type] 2-word spindle
[Unit of data] Detection unit

- **Spindle-converted position deviation difference during rigid tapping (momentary value)**

0456	Spindle-converted position deviation difference during rigid tapping (momentary value)

[Data type] 2-word spindle
[Unit of data] Detection unit

- **Synchronization error range during rigid tapping (momentary value)**

0457	Synchronization error range during rigid tapping (maximum value)

[Data type] 2-word spindle
[Unit of data] Detection unit

- **Spindle-converted move command difference during rigid tapping (maximum value)**

0460	Spindle-converted move command difference during rigid tapping (maximum value)

[Data type] 2-word spindle
[Unit of data] Detection unit

- **Spindle-converted machine position difference during rigid tapping (momentary value)**

0461	Spindle-converted machine position difference during rigid tapping (momentary value)

[Data type] 2-word spindle
[Unit of data] Detection unit

- **Spindle-converted machine position difference during rigid tapping (maximum value)**

0462	Spindle-converted machine position difference during rigid tapping (maximum value)

[Data type] 2-word spindle
[Unit of data] Detection unit

10.13.13 Command Format

T

Command format for the T series

The rigid tapping mode can be specified in one of two formats: FS0 format and FS10/11 format. A usable format can be selected by parameter setting.

Parameter		Executable command format
FCV (No.0001#1)	F0C (No.5102#3)	
0	-	FS0 format only (The number of repeats is specified using address K.)
1	0	FS10/11 format or FS0 format (The number of repeats is specified using address K.)
1	1	FS0 format only (The number of repeats is specified using address L.)

This subsection describes the FS0 command format for rigid tapping on the T series. For the M series, refer to the description of "RIGID TAPPING" in the Operator's Manual for machining center system.

- G84 : Front tapping cycle

The first axis of a plane is a drilling axis, and the other axes are positioning axes.

Parameter RTX(No.5209#0)	Plane selection	Drilling axis
0	G17 Xp-Yp plane	Xp
	G18 Zp-Xp plane	Zp
	G19 Yp-Zp plane	Yp
1 (Note)		Zp

Xp : X axis or its parallel axis
Yp : Y axis or its parallel axis
Zp : Z axis or its parallel axis

Note) These are invalid in the FS10/11 format.

1. When the parameter FCV (No.0001#1) is set to 0 :
   G84 X_C_ Z_ R_ P_ F_ K_ M_ ;
   - G84 : Front tapping cycle G code
   - X C : Tapping position
   - Z : Position of the bottom face of a tapped hole
   - R : Position of an R point (incremental command at all times)
   - P : Dwell time at the bottom of a tapped hole
   - F : Cutting feedrate
   - K : Number of repeats
   - M : M code for C-axis clamping

2. When the parameter FCV (No.0001#1) is set to 1 :
   G84 X_C_ Z_ R_ P_ F_ L_ M_ ;
   - G84 : Front tapping cycle G code
   - X C : Tapping position
   - Z : Position of the bottom face of a tapped hole
   - R : Position of an R point (incremental command at all times)
   - P : Dwell time at the bottom of a tapped hole
   - F : Cutting feedrate
   - L : Number of repeats
   - M : M code for C-axis clamping

- G88 : Side tapping cycle

The second axis of a plane is a drilling axis, and the other axes are positioning axes.

Parameter RTX(No.5209#0)	Plane selection	Drilling axis
0	G17 Xp-Yp plane	Yp
	G18 Zp-Xp plane	Xp
	G19 Yp-Zp plane	Zp
1 (Note)		Xp

Xp : X axis or its parallel axis
Yp : Y axis or its parallel axis
Zp : Z axis or its parallel axis

Note) These are invalid in the FS10/11 format.

1. When the parameter FCV (No.0001#1) is set to 0 :
   G88 Z_C_ X_ R_ P_ F_ K_ M_ ;
   - G88 : Side tapping cycle G code
   - Z C : Tapping position
   - X : Position of the bottom face of a tapped hole
   - R : Position of an R point (incremental command at all times)
   - P : Dwell time at the bottom of a tapped hole
   - F : Cutting feedrate

K : Number of repeats
M : M code for C-axis clamping

2. When the parameter FCV (No.0001#1) is set to 1 :
G88 Z_C_ X_ R_ P_ F_ L_ M_ ;
G88 : Side tapping cycle G code
Z C : Tapping position
X : Position of the bottom face of a tapped hole
R : Position of an R point (incremental command at all times)
P : Dwell time at the bottom of a tapped hole
F : Cutting feedrate
L : Number of repeats
M : M code for C-axis clamping

- Method of specification

The rigid tapping mode can be specified using one of three methods:

- Specification of M29S_ before specifying a tapping cycle
- Specification of M29S_ in the same block
- Enabling rigid tapping to be performed without specifying M29S_

When using the third method, specify S_ either before or in a block containing G84 (G88).
Thus, the spindle stops, after which the tapping cycle specified next is placed in rigid tapping mode.

- Specifying M29 before a block containing G84 (G88)

```
M29 S_;
G84(G88) X_C_(Z_C_) Z_(X_) R_ P_ F_ K_ M_;
X_C_(Z_C_);
X_C_(Z_C_);
    :
G80;
```

Rigid tapping mode

- Specifying M29 and G84 (G88) in the same block
(However, the M code for C-axis clamping cannot be specified.)

```
G84(G88) X_C_(Z_C_) Z_(X_) R_ P_ F_ K_ M29 S_;
X_C_(Z_C_);
X_C_(Z_C_);
    :
G80;
```

Rigid tapping mode

- Converting G84 (G88) to a G code for rigid tapping
(by setting bit 0 (G84) of parameter No. 5200 to 1)

```
G84(G88) X_C_(Z_C_) Z_(X_) R_ P_ F_ K_ S_ M_;
X_C_(Z_C_);
X_C_(Z_C_);
    :
G80;
```

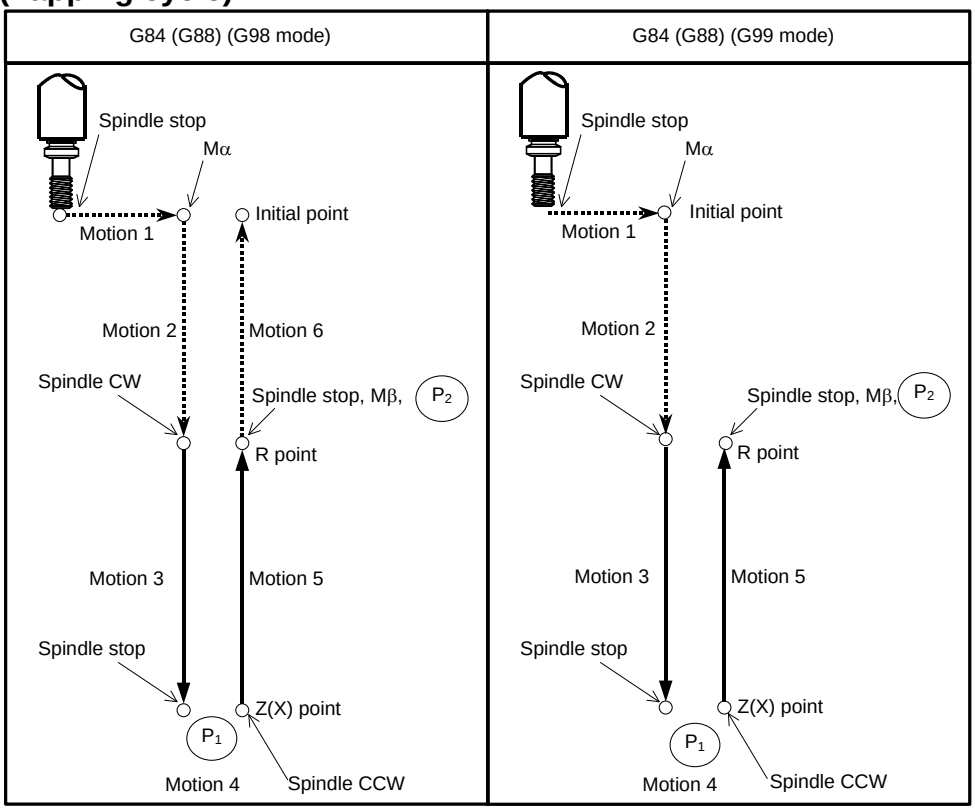
Rigid tapping mode

- Notes on the lathe system

NOTE
1 In feed per minute mode, F_/S**** determines a thread lead. In feed per rotation mode, F_ specifies a thread lead.
2 S**** must specify a value that does not exceed the value set in the maximum spindle speed parameter (No. 5241 to 5244) for the gear to be used. Otherwise, an alarm PS0200 is issued in a block containing G84 (G88).

NOTE
3 F_ must specify a value that does not exceed the maximum cutting feedrate. When 0 is specified, an alarm PS0201 is issued.
4 Between M29 and G84 (G88), S and a command for movement along an axis must not be specified. Further, M29 must not be specified in a tapping cycle. Otherwise, alarms PS0203 and PS0204 are issued, respectively.
5 In the rigid tapping mode, G84 and G88 must not be specified in succession. Cancel the rigid tapping mode before specifying G84 and G88 in succession.

G84, G88 (Tapping cycle)



⚠ CAUTION
For cutting feed along the Z(X)-axis and override in extraction, see Subsection "Override".

NOTE
G code system A does not include G98 (return to initial level) and G99 (return to R point level). Return to the initial level is always used.

10.13.14 Position Control Loop Gain Parameter Switching

In rigid tapping, the position control loop gain of the drilling axis must match that of the spindle. Specifically, change the position control loop gain of the drilling axis at the following timings:

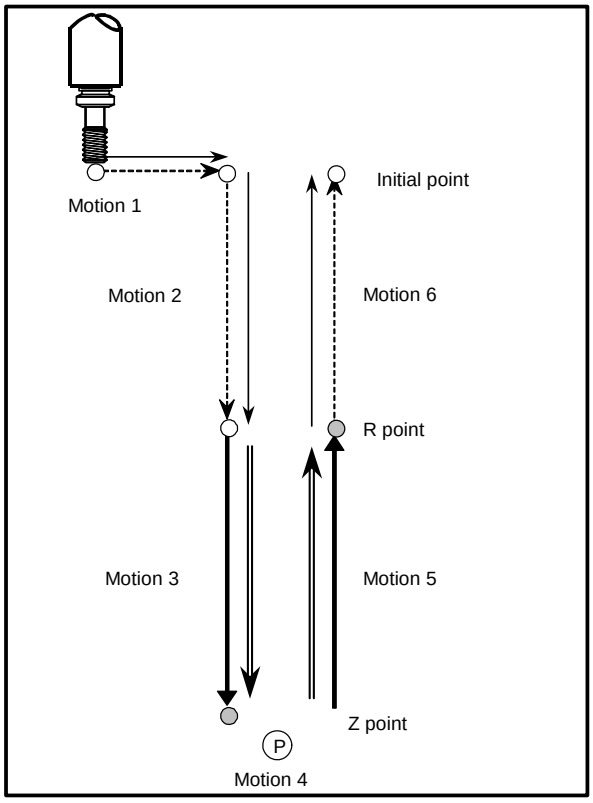
(1) A change from a position control loop gain for normal operation to a position control loop gain for rigid tapping must be made when the cutting along the drilling axis is started (when operation 3 is started).
(2) A change from a position control loop gain for rigid tapping to a position control loop gain for normal operation must be made when the operation after the completion of extraction along the drilling axis is started (when operation 6 is started).

While a position control loop gain for rigid tapping is selected, the parameters dedicated to rigid tapping are used for the following parameters for the drilling axis:

(1) In-position width
(Parameters No.5300 and 5302)
(2) Position deviation limit during stop
(Parameters No.5312 and 5352)
(3) Position deviation limit during travel
(Parameters No.5310 and 5350)

The above parameters dedicated to rigid tapping are usually used to specify the in-position width at the R point for operation 2. However, an ordinary parameter (No.1826) can be selected by using bit 4 (IRR) of parameter No. 5202.

NOTE
If bit 5 (NCI) of parameter No. 1601 is set to "1", no in-position check is made at the R point for operation 2 except when bit 1 (RIP) of parameter No. 5209 is set to "1".



1) As in-position width parameters:
○ Ordinary parameters are used.
● Parameters for rigid tapping are used.

2) As parameters for position deviation limits during stop:
○ Ordinary parameters are used.
● Parameters for rigid tapping are used.

3) As parameters for position deviation limits during travel:
→ Ordinary parameters are used.
⇒ Parameters for rigid tapping are used.

---------- Rapid traverse
——— Z axis feed
(P) Dwell

10.13.15 Signal

10.13.15.1 Signals for the rigid tapping function

Rigid tapping signal RGTAP<Gn061.0>

[Classification] Input signal

[Function] When M29 (miscellaneous function for preparation for rigid tapping) is specified, the PMC enters rigid tapping mode, then turns on this signal to notify the CNC.

1 : The PMC enters in rigid tapping mode.

0 : The PMC does not enter rigid tapping mode.

For an explanation of placing the PMC in rigid tapping mode, see the description of the interface with the PMC, given later.

This signal posts whether the PMC has entered rigid tapping mode. If this signal is not set to 1, even when M29 is specified, a P/S alarm is issued in a G84/G74(M series) or G84/G88(T series) block.

Spindle rotation direction signals RGSPM,RGSPP<Fn065.1,0>

[Classification] Output signal

[Function] During rigid tapping, these signals notify the PMC of whether the spindle is rotating in the forward or reverse direction.

→ During rigid tapping, the spindle is:

RGSPP 1 : Rotating in the forward direction (CW).

0 : Not rotating in the forward direction (CW).

RGSPM 1 : Rotating in the reverse direction (CCW).

0 : Not rotating in the reverse direction (CCW).

[Output cond.] These signals are output when the spindle is rotating in rigid tapping mode. This means that, even in rigid tapping mode, these signals are not output, for example, when the spindle is being positioned to a hole position, or a dwell operation is in progress at the bottom of a hole or at an R point.

These signals are not output in the feed hold state or single block stop state. When the spindle is placed in the interlock stop state, machine lock state, or Z-axis ignore state, however, the spindle is not regarded as having stopped; these signals are output.

These signals are valid only in rigid tapping mode. In normal spindle control, these signals are not output; both RGSPP and RGSPM are set to "0".

Rigid tapping in-progress signal RTAP<Fn076.3>

[Classification] Output signal

[Function] This signal notifies the PMC that rigid tapping mode is active.

RTAP 1 : Rigid tapping mode is currently active.

0 : Rigid tapping mode is not currently active.

By latching M29, the PMC knows that rigid tapping mode has been specified, and thus performs the required processing on the PMC side. This signal can substitute for the latching of M29. Even in this case, however, FIN for M29 cannot be omitted.

10.13.15.2 Signals related to S code output

Spindle enable signal ENB<Fn001.4>

[Classification] Output signal

[Function] These signals post whether the spindle output is 0. In rigid tapping mode, these signals are used to cancel rigid tapping in a PMC sequence associated with rigid tapping. For details, see the explanation of the interface with the PMC, given later.

Spindle-speed function code signals (binary output) S00 to S31<Fn022 to Fn025>
Spindle-speed function strobe signals SF<Fn007.2>

[Classification] Output signal

[Function] These signals send S codes specified for the CNC, in binary format, to the PMC.

[Output cond.] When an S code is specified, the specified value is output, in binary format, with these signals. When the new spindle speed data is fully set, the SF signal is set to "1".
Before rigid tapping can be performed, however, parameter setting is required to output these signals, as described below.

M series :

SF output depends on the gear selection method, as described below.

[1] M-type gear selection method
SF output depends on bit 6 (SFA) of parameter No. 3705.

[2] T-type gear selection method
SF output depends on the setting of bit 5 (NSF) of parameter No. 3705.

T series :

The following parameter needs to be set to output S codes and SF:
Bit 4 (EVS) of parameter No. 3705 is set to 1

In rigid tapping, when SF is to be used by the PMC to read an S code output signal for gear switching or output switching, set the above parameters as required.

NOTE
1 The timing charts, given later, give examples of gear switching by setting the parameters as follows:
M series : SFA=0,NSF=0
T series : EVS=1
2 When the constant surface speed control function is being used, an S code (specifying a surface speed) used for constant surface control (G96) is output. Such an S code can be distinguished from an S code used for specifying a rotation speed. One method is to use, for example, the constant surface speed control in-progress signal <F002.2> for the processing performed on the PMC side. Another method is to mask the S code and SF signal, output by setting bit 0 (ESF) of parameter No. 3705.

10.13.15.3 Signals related to gear switching

Gear selection signals (output) GR3O,GR2O,GR1O<Fn034.2,1,0>

M

[Classification] Output signal

[Operation] When M-type gear selection is being used, these signals are used in a PMC sequence for rigid tapping.
The signals post, to the PMC, information about a spindle gear to be used, according to the value of S_ specified at the execution of G84 (G74).
When gear switching becomes necessary, the states of the signals change together with the SF signal.
The PMC should perform gear switching according to the information posted by the gear selection signals.
Reference information:

The table below indicates the relationship between the output signals and gear selection.

	GR3O	GR2O	GR1O
1st (low) speed gear	0	0	1
2nd (medium) speed gear	0	1	0
3rd (high) speed gear	1	0	0

Gear selection signals (input) GR2,GR1<Gn028.2,1>

[Classification] Input signal

[Operation] When T-type gear selection is being used, these signals are used in a PMC sequence for rigid tapping.

The signals post, to the CNC, information about a spindle gear to be used.

Reference information:

The table below shows the relationship between the output signals and spindle gear selection.

	GR1	GR2
1st (low) speed gear	0	0
2nd (medium) speed gear	1	0
3rd (high) speed gear	0	1
4th (high) speed gear	1	1

The serial spindle clutch/gear selection signals (CTH1A, CTH2A, CTH1B, CTH2B) must be set in addition to the setting of the gear selection signal described above.

NOTE

In M series rigid tapping, the specification of the 4th (high) speed gear is invalid. If specified, the system assumes that the 3rd (high) speed gear has been specified.

10.13.15.4 Signals related to the addition of multi spindle control

Spindle enable signals ENB<Fn001.4>,ENB2<Fn038.2>

[Classification] Output signal

[Function] These signals post whether the spindle output to each spindle is 0 in multi spindle control. In rigid tapping mode, these signals are used to cancel rigid tapping in a PMC sequence associated with rigid tapping.

For details, see the explanation of the interface with the PMC, given later.

Gear selection signals (input) GR2,GR1<Gn028.2,1>,GR22,GR21<Gn029.1,0>,

[Classification] Input signal

[Operation] When the T-type gear selection method is employed in multi spindle control, these signals are used with a PMC sequence for rigid tapping.

The signals post, to the CNC, information about a spindle gear to be used.

Reference information:

The table below shows the relationship between the output signals and spindle gear selection.

	GRs1	GRs2
1st (low) speed gear	0	0
2nd (medium) speed gear	1	0
3rd (high) speed gear	0	1
4th (high) speed gear	1	1

(s is -, 2.)

The serial spindle clutch/gear selection signals (G070.3,2 for the first spindle, G074.3,2 for the second spindle) must be set in addition to the setting of the gear selection signal described above.

NOTE
In M series rigid tapping, the specification of the 4th (high) speed gear is invalid. If specified, the system assumes that the 3rd (high) speed gear has been specified.

Spindle selection signals SWS1<Gn027.0>,SWS2<Gn027.1>
Rigid tapping spindle selection signals RGTSP1<Gn061.4>,RGTSP2<Gn061.5>

[Classification] Input signal

[Operation] SWS1 to SWS2 are used to transfer spindle commands when the multi-spindle control is used. In rigid tapping, the signals can be shared to select a spindle to be used for rigid tapping. (The signals can be used for this purpose when bit 7 (SRS) of parameter No. 5200 is set to 0.)
RGTSP1 and RGTSP2 are used to select a spindle used for rigid tapping, independently of the SWS1 to SWS2 signals, when the multi-spindle control is being used. (The signals can be used when bit 7 (SRS) of parameter No. 5200 is set to 1. These signals are supported only by the T series.)
See the tables below for details of the settings of these signals.
When bit 7 (SRS) of parameter No. 5200 is set to 0, to select a spindle to be used for rigid tapping, set the signals as indicated below.

Spindle used for rigid tapping	Signal state	
	SWS1	SWS2
First spindle	1	0
Second spindle	0	1
Alarm PS0205 is issued.	0	0

When bit 7 (SRS) of parameter No. 5200 is set to 1, to select a spindle to be used for rigid tapping, set the signals as indicated below.

Spindle used for rigid tapping	Signal state	
	RGTSP1	RGTSP2
First spindle	1	0
Second spindle	0	1
Alarm PS0205 is issued.	0	0

NOTE
1 These signals must be applied before the command for rigid tapping (M29 S....; G84 X...) is specified. The states of these signals must not be changed before rigid tapping has been completed.
2 When multiple signals from SWS1 to SWS2 are set to 1 simultaneously, the signals are checked in the order from SWS1 to SWS2, and the spindle corresponding to the signal that is first found to be set to 1 is assumed to be specified.
3 Similarly, when multiple signals from RGTSP1 to RGTSP2 are set to 1 simultaneously, the signals are checked in the order from RGTSP1 to RGTSP2, and the spindle corresponding to the signal that is first found to be set to 1 is assumed to be specified.

Spindle-by-spindle stop signals *SSTP1<Gn027.3>,*SSTP2<Gn027.4>

[Classification] Input signal

[Operation] These signals are used to stop each spindle when the multi-spindle control is used. In a PMC sequence for rigid tapping, the ENB and ENB2 signals are used. Accordingly, the logic of the signals used for a spindle selected to perform rigid tapping must match the logic of the spindle stop signal *SSTP.

*SSTP1 1 : The output to the first spindle is not forced to 0 min^{-1}.
0 : 0 min^{-1} is commanded to first spindle.

*SSTP2 1 : The output to the second spindle is not forced to 0 min^{-1}.
0 : 0 min^{-1} is commanded to second spindle.

Position coder selection signal PC2SLC<Gn028.7>

[Classification] Input signal

[Operation] This signal is used to select the position coder when the multi-spindle control is being used. Note, however, that it cannot be used with a spindle selected to perform rigid tapping.
For rigid tapping, this signal is not used. Instead, a position loop is constructed by combining the first spindle with the first position coder or by combining the second spindle with the second position coder.
However, the display of the actual speed is switched by this signal, even during rigid tapping.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn027				*SSTP2	*SSTP1		SWS2	SWS1
Gn028	PC2SLC					GR2	GR1	
Gn029		*SSTP					GR22	GR21
Gn031			GR42	GR41				
Gn061			RGTSP2	RGTSP1				RGTAP

	#7	#6	#5	#4	#3	#2	#1	#0
Fn001				ENB				
Fn007						SF		
Fn034						GR3O	GR2O	GR1O
Fn022	S07	S06	S05	S04	S03	S02	S01	S00
Fn023	S15	S14	S13	S12	S11	S10	S09	S08
Fn024	S23	S22	S21	S20	S19	S18	S17	S16
Fn025	S31	S30	S29	S28	S27	S26	S25	S24
Fn038						ENB2		
Fn065							RGSPM	RGSPP
Fn076					RTAP			

10.13.15.5 Notes on interface with the PMC

The following describes some notes in designing the interface with the PMC.

- Rigid tapping mode management and spindle enable signals (ENB, ENB2)

In the PMC-based rigid tapping mode management, enter the rigid tapping mode by using M29, and cancel the mode on the falling edge of a spindle enable signal (ENB or ENB2) during a reset or in the rigid tapping mode. In rigid tapping, spindle enable signals are used in this way, so the spindle stop signal (*SSTP) must not be set to "0".
However, spindle orientation (*SSTP or SOR) is sometimes used for gear change. In this case, the PMC must be controlled not to cancel the rigid tapping mode on a falling edge of ENB if *SSTP is set to "0". Instead of M29 and the ENB signal, the RTAP signal may be used; the rigid tapping mode may be set on the rising edge of the RTAP signal, and the mode may be canceled on the falling edge of the RTAP signal.
When rigid tapping is performed under multi-spindle control, spindle enable signal ENBs for the spindle with which rigid tapping is performed must be used for managing the rigid tapping mode.

- Controlling spindle output by the PMC

When the SIND signal is set to "1", spindle output is controlled by the signals (SSIN, SGN, R1I to R12I) output from the PMC.
At this time, the effect of ENB is as described above. In addition, when rigid tapping mode is canceled in a block containing G80;, the momentary rotation of the spindle, caused by a delay in the PMC processing, can result. Accordingly, the PMC's control over spindle output must be disabled in rigid tapping mode by setting SIND to "0".
In rigid tapping under multi spindle control, ensure, for a similar reason, that in the rigid tapping mode, PMC-based control is disabled (SINDs = "0") for the spindle used for rigid tapping.

- T-type gear selection method

When T-type gear selection is used, the PMC must determine whether gear switching is to be performed, and subsequently perform gear switching as required. For this purpose, each time a spindle-speed function code is specified, the spindle-speed function code read signal (SF) and spindle-speed function code signals (S00 to S31) must be output to the PMC. The required parameter settings are described below.

- M series :
  Set bit 5 (NSF) of parameter No. 3705 to 0 to output SF.
- T series :
  Set bit 4 (EVS) of parameter No. 3705 to 1 to output SF.

- Gear switching timing

In general, a block containing M29 (miscellaneous function for preparation for rigid tapping) specifies S_, S_ being output when a block containing G84/G74 (M series) or G84/G88 (T series) is executed. This means that gear switching is performed in the block specifying G84 (G74).

- When rigid tapping mode is specified

M29 (miscellaneous function for preparation for rigid tapping) and S_ specify rigid tapping mode. When M29 is accepted by the PMC, the following processing must be performed:

- Stop the spindle when it is rotating.
- Check that the spindle has stopped completely, then set the rigid tapping signal RGTAP <G061.0> to on.
- Activate the spindle motor. Activate the motor so that a positive speed command rotates the spindle in the forward direction (CCW when viewed from the - side of the tapping axis).
- Return FIN at least 250 ms after activation.

NOTE
The condition "at least 250 ms after activation" results from there being no way of checking the completion of spindle motor activation. Therefore, this wait period serves as an alternative. The time required for activation to be completed varies with the spindle motor and amplifier. Therefore, this value of 250 ms is given as a guideline only.

In an M29 block, S_ is not executed, merely being read in. Spindle output is equivalent to the specification of S0. S_ is executed in a G84/G74 (M series) or G84/G88 (T series.) The timing chart is shown in the chart indicating the execution of G84/G74 or G84/G88.

- Execution of G84/G74 (M series) or G84/G88 (T series) block

When M29S_; is specified, S_ is read in, spindle output being equivalent to the specification of S0;. S_ is output when G84/G74 (M series) or G84/G88 (T series) block is executed. Thus, the processing described below is performed.

- When M-type gear selection is used

When using a machine that features multiple gear stages for use with the spindle motor and spindle, and the newly programmed S_ is outside the previously selected gear range, the spindle-speed function strobe signal SF <F007.2> and gear selection signals (output) GR3O, GR2O, GR1O <F034.2, 1, 0> are output to the PMC.
At this time, perform gear switching at the PMC.

- When T-type gear selection is used

The spindle-speed function strobe signal SF <F007.2> and spindle-speed function code signals S00 to S31 <F022 to F025> are output to the PMC. (However, parameter setting is required to enable output of the S codes and SF signal. See the description of each bit of parameter No. 3705.)
At this time, the PMC must determine whether gear switching is to be performed, and perform gear switching as required. The selected gear must be reflected in the gear selection signals (input) GR2 and GR1 <G028.2, 1> for notification to the CNC.
From GR2 and GR1, the CNC determines which gear is selected.
However, note the difference between the M series and T series, as described below.

- M series :
  Up to three gear stages are supported. If the fourth gear stage is selected, it is assumed that the third gear stage is selected.
- T series :
  Up to four gear stages are supported.
  (A gear selection is posted to the CNC with the gear selection signal for each spindle.)

- C-axis clamp/unclamp

T

When an M code (parameter No.5110) for C-axis clamping is specified in a block specifying G84/G88, the M codes for C-axis clamping/unclamping are output at the following timings:

- The M code for clamping is output at the time of rapid traverse to the R point level (operation 2). When FIN is returned, tapping (operation 3) starts.
- After extraction from the hole bottom to the R point level (operation 5), the M code for unclamping is output. When FIN is returned, dwell or rapid traverse to the initial level (operation 6) starts.

The rigid tapping mode can be specified in three different methods. One method specifies M29 before G84. The second method specifies M29 and G84 in the same block. The third method uses G84 as a rigid tapping G code. In any method, however, PMC processing is the same. (In any case, the M29 code is output.)

10.13.16 Timing Charts for Rigid Tapping Specification

The timing chart for rigid tapping specification depends on the method used to specify rigid tapping mode, the gear selection method (M-type or T-type), and whether to perform gear switching.
From the table, find the appropriate timing chart and apply the information it contains as necessary.

M

Gear selection method M-type T-type	Gear switching	Specification method		
		M29 is specified before G84/G74.	M29 and G84/G74 are specified in the same block.	By parameter setting, G84/G74 is specified as a G code for rigid tapping.
M-type	Not performed	Fig.10.12.16(a)	Fig.10.12.16(e)	Fig.10.12.16(i)
	Performed	Fig.10.12.16(b)	Fig.10.12.16(f)	Fig.10.12.16(j)
T-type	Not performed	Fig.10.12.16(c)	Fig.10.12.16(g)	Fig.10.12.16(k)
	Performed	Fig.10.12.16(d)	Fig.10.12.16(h)	Fig.10.12.16(l)

T

Gear switching	Specification method		
	M29 is specified before G84/G88.	M29 and G84/G88 are specified in the same block.	By parameter setting, G84/G88 is specified as a G code for rigid tapping.
Not performed	Fig.10.12.16(m)	Fig.10.12.16(o)	Fig.10.12.16(q)
Performed	Fig.10.12.16(n)	Fig.10.12.16(p)	Fig.10.12.16(r)

NOTE
For more information about the M/T type gear selection method, see Section "SPINDLE SPEED CONTROL." Note the following:
M series
when constant surface speed control is not used and bit 4 (GTT) of parameter No. 3706 is set to 0. → M-type
When constant surface speed control is used, or bit 4 (GTT) of parameter No. 3706 is set to 1.
→ T-type
T series :
T-type only

10.13.16.1 When M29 is specified before G84/G74

- **M type gear selection method**

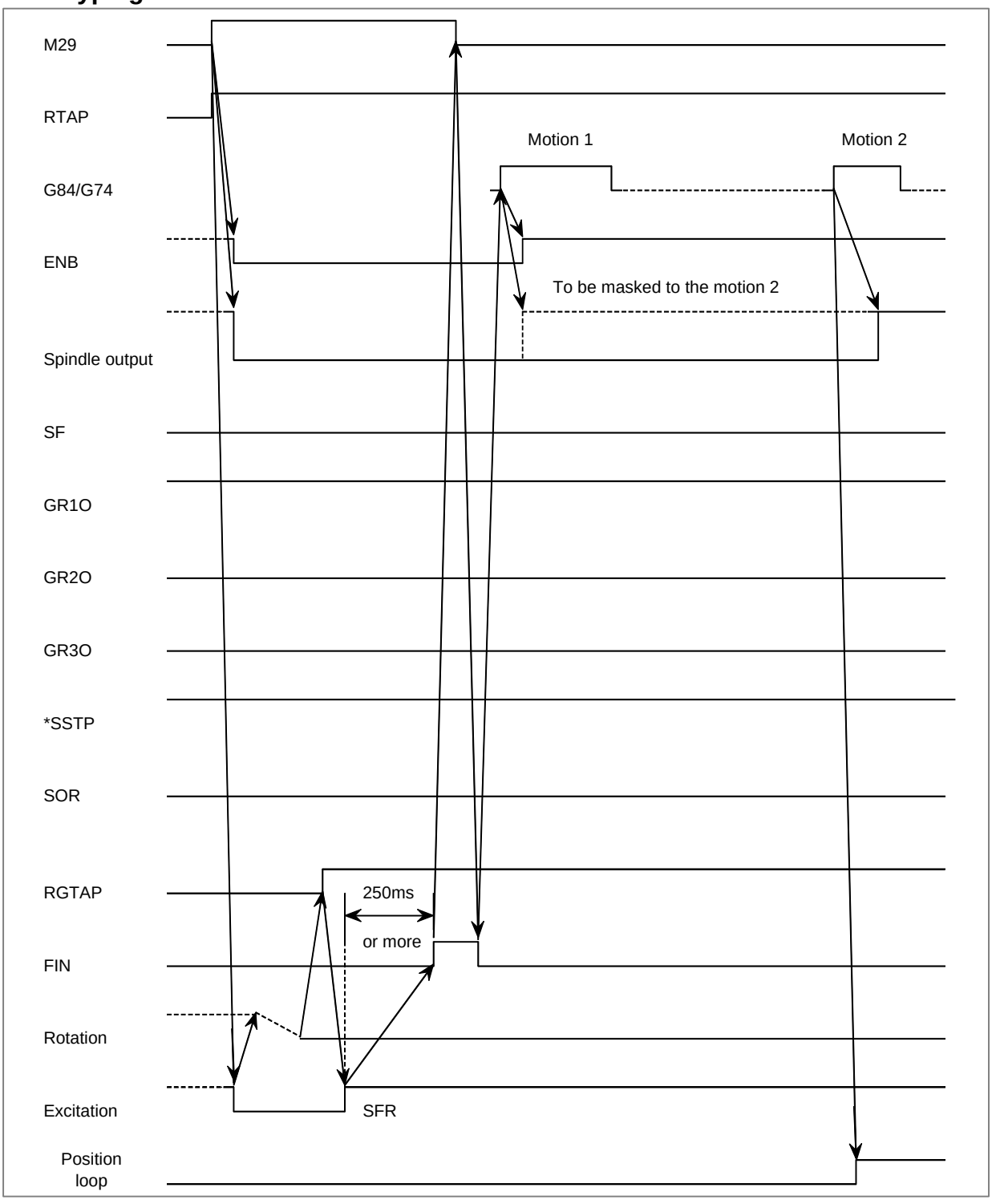


Fig. 10.13.16 (a) Gear is not changed

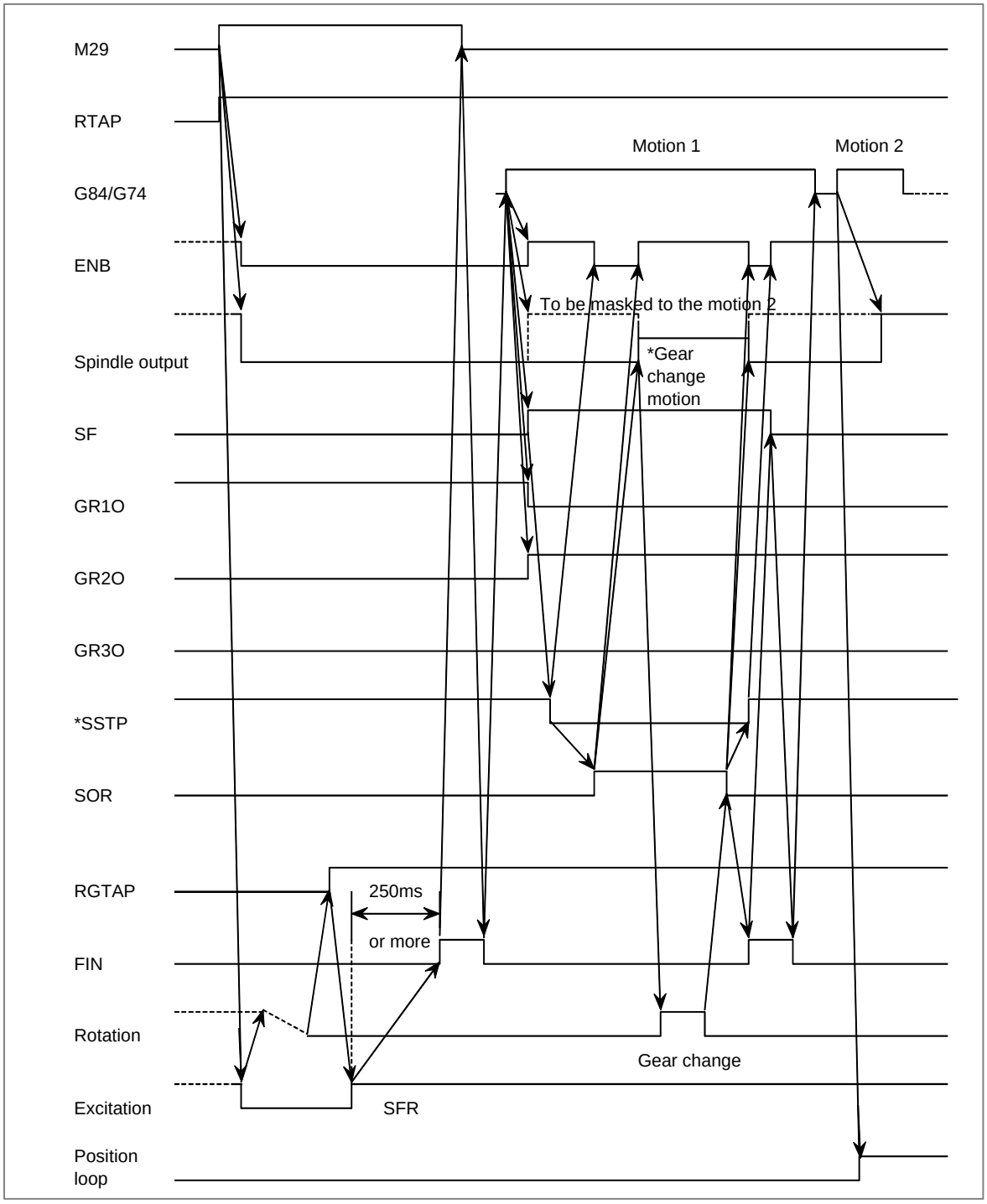


Fig. 10.13.16 (b) When gear change is performed (from low to middle gear)

NOTE

This time chart show an example where the gear has shifted from low to middle gear. One of the gear select signals (GR1O, GR2O, GR3O) has turned from "1" to "0", and one of the two remaining signals has turned from "0" to "1". This changes the gear.

- **T type gear selection method**

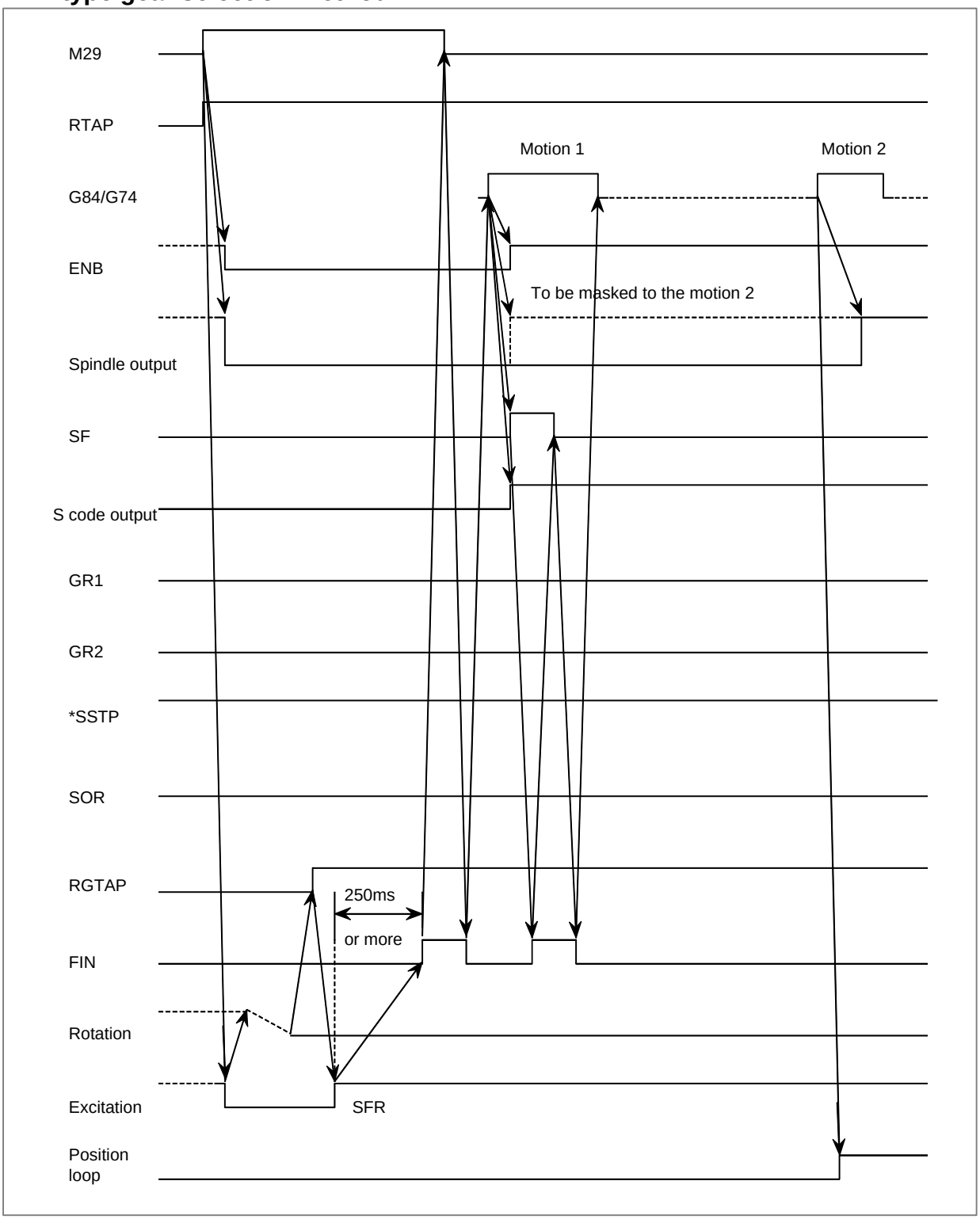


Fig. 10.13.16 (c) Gear change is not performed

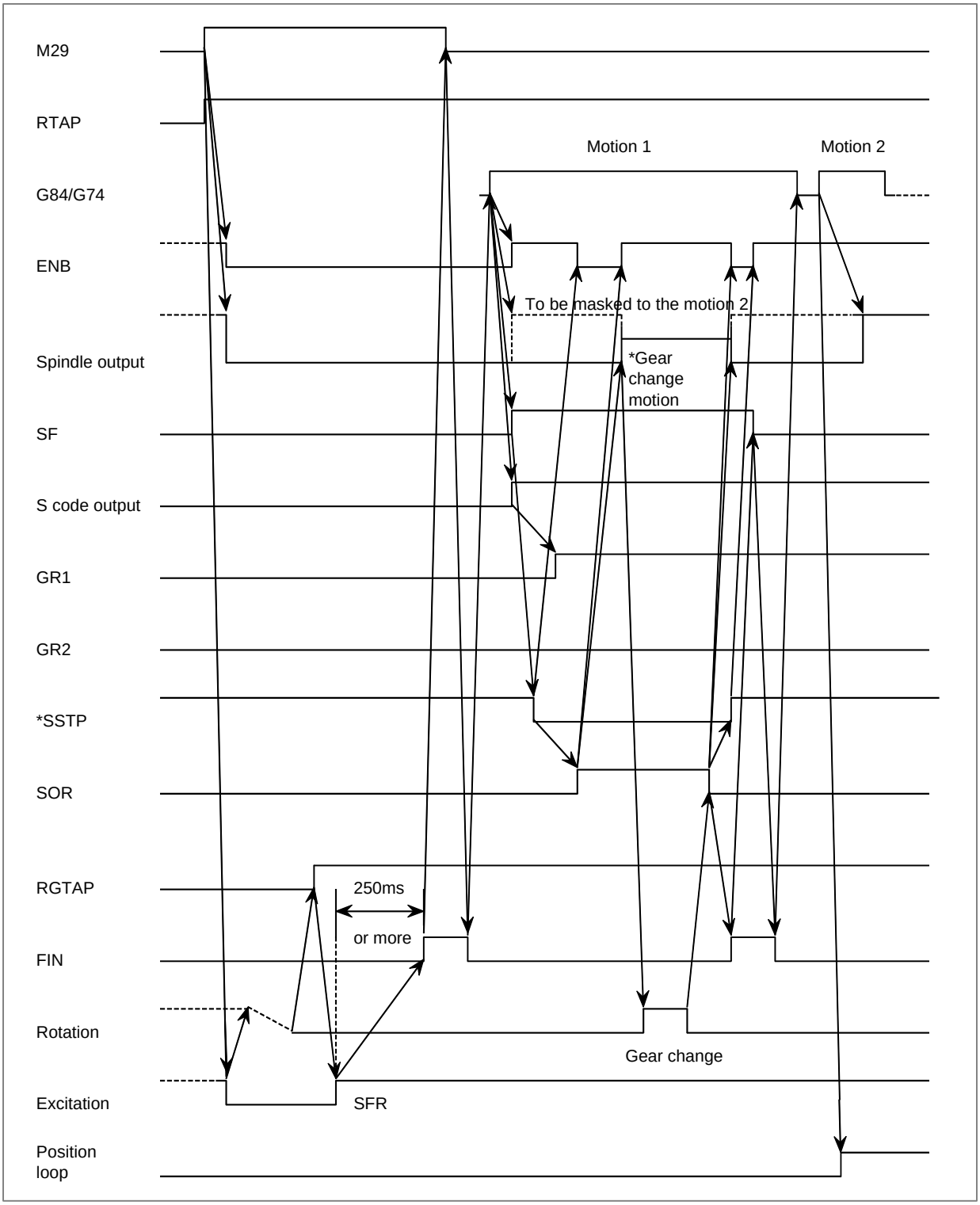


Fig. 10.13.16 (d) When gear-change is performed (low to middle gear)

NOTE

This time chart shows an example where the gear has changed from low to middle gear. The PMC selects a required gear through an S code output and selects the gear using the GR1 and GR2 signals to inform CNC of the selected gear.

10.13.16.2 M29 and G84/G74 are specified in the same block

- **M type gear selection**

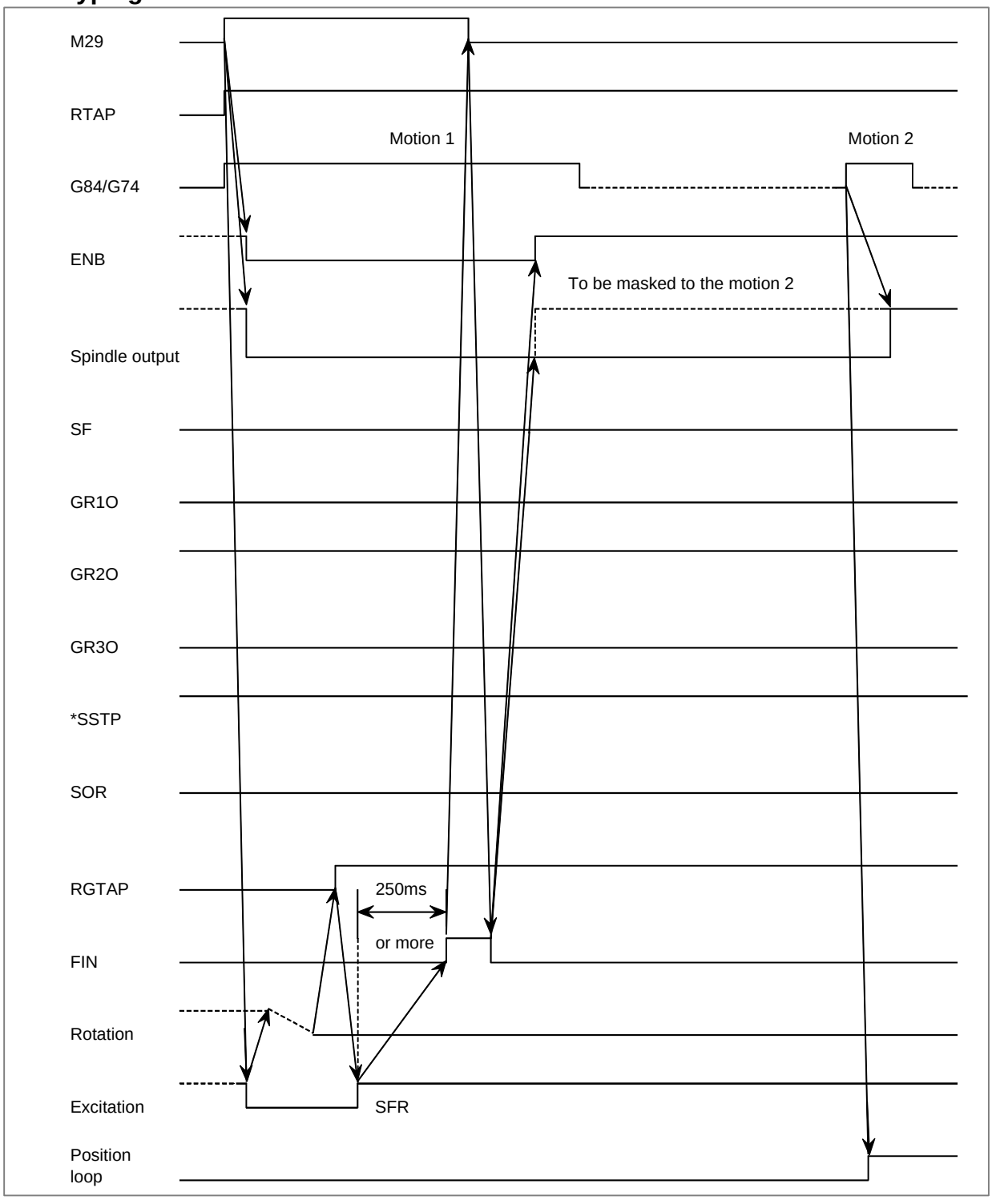


Fig. 10.13.16 (e) When gear-change is not performed

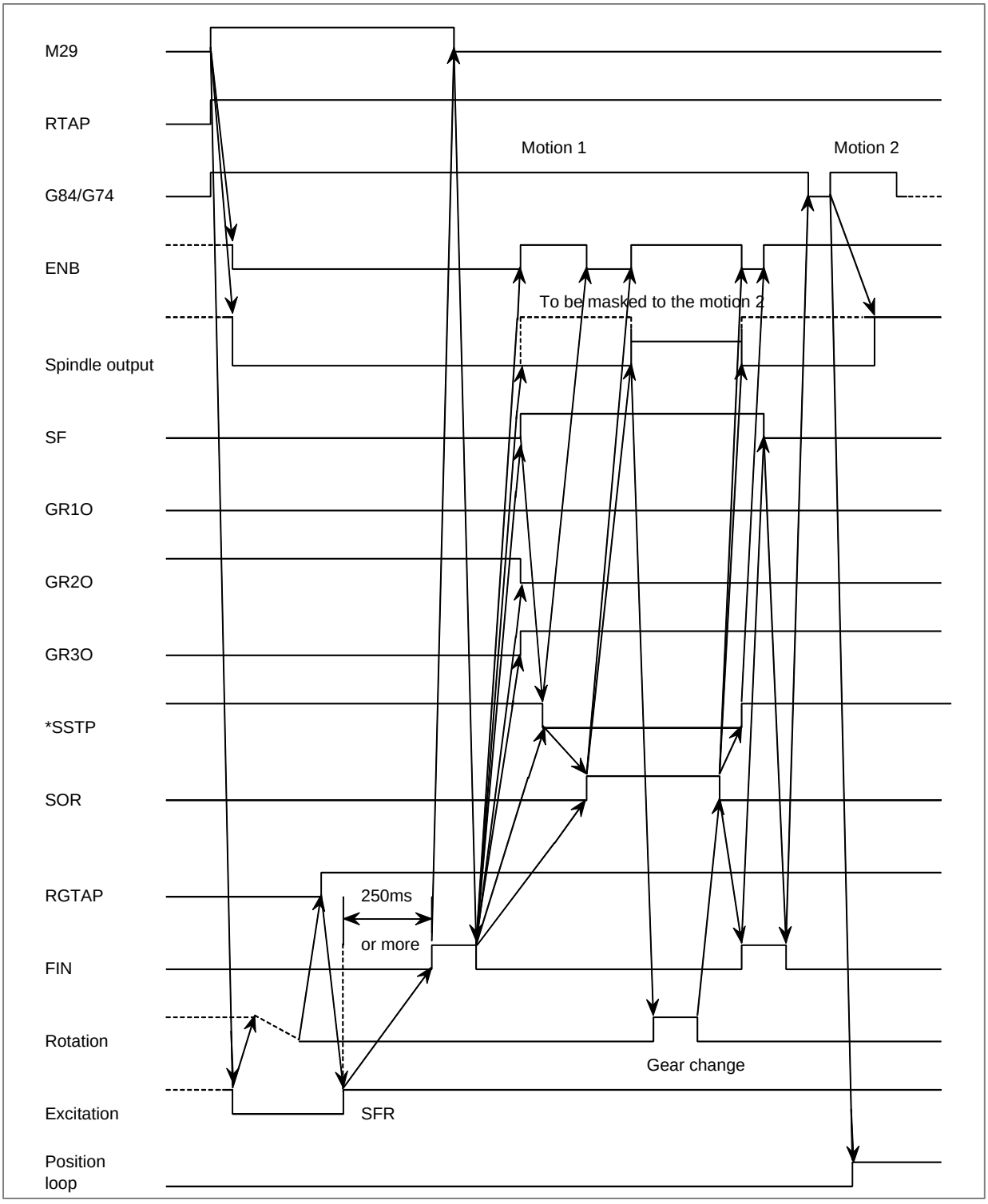


Fig. 10.13.16 (f) When gear-change is performed (middle to high)

NOTE

This time chart shows an example where the gear has shifted from middle high gear. One of the gear select signals (GR1O, GR2O, GR3O) has turned from "1" to "0", and one of the two remaining signals has tuned from "0" to "1". This changes the gear.

- T type gear selection method

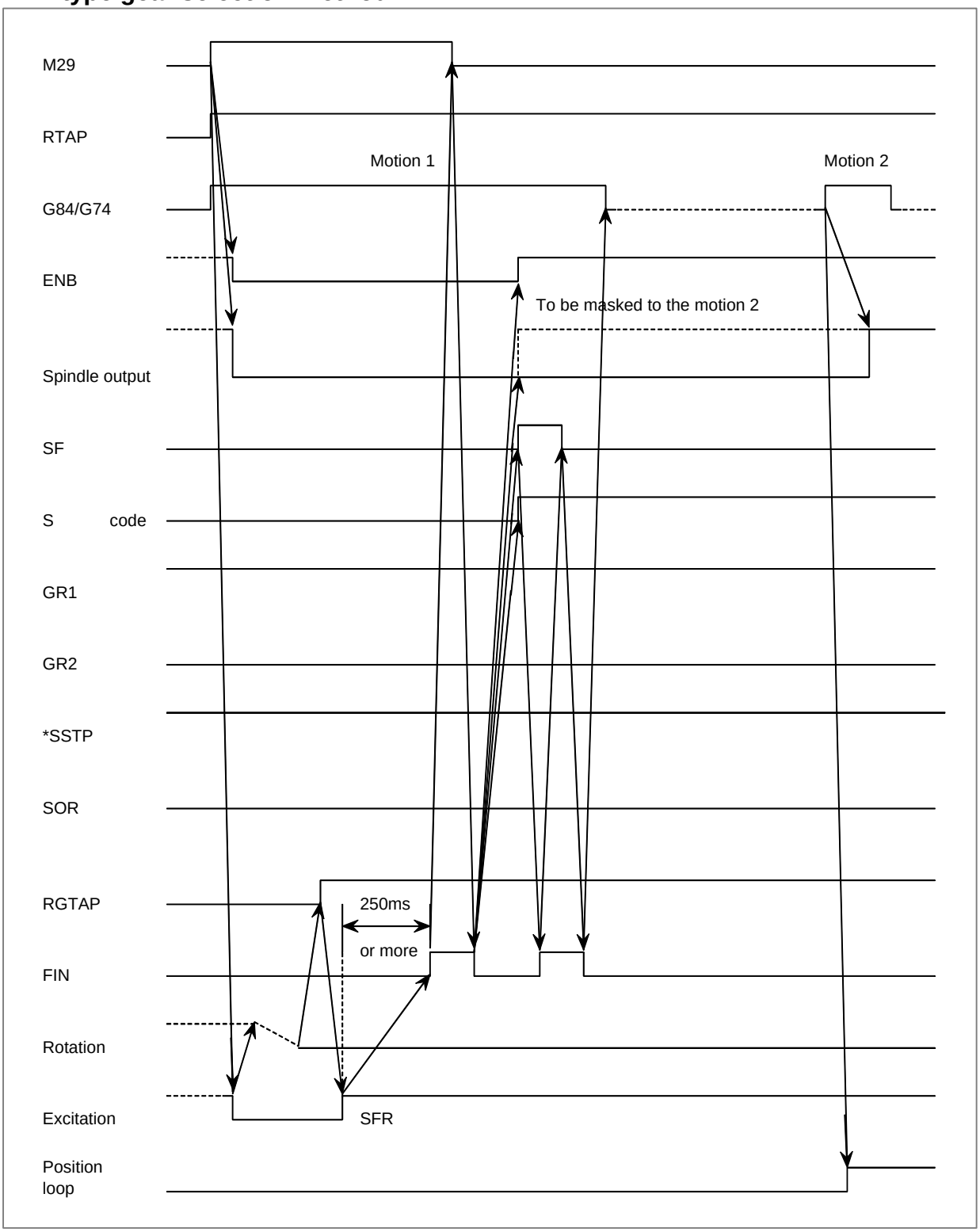


Fig. 10.13.16 (g) When gear change is not performed

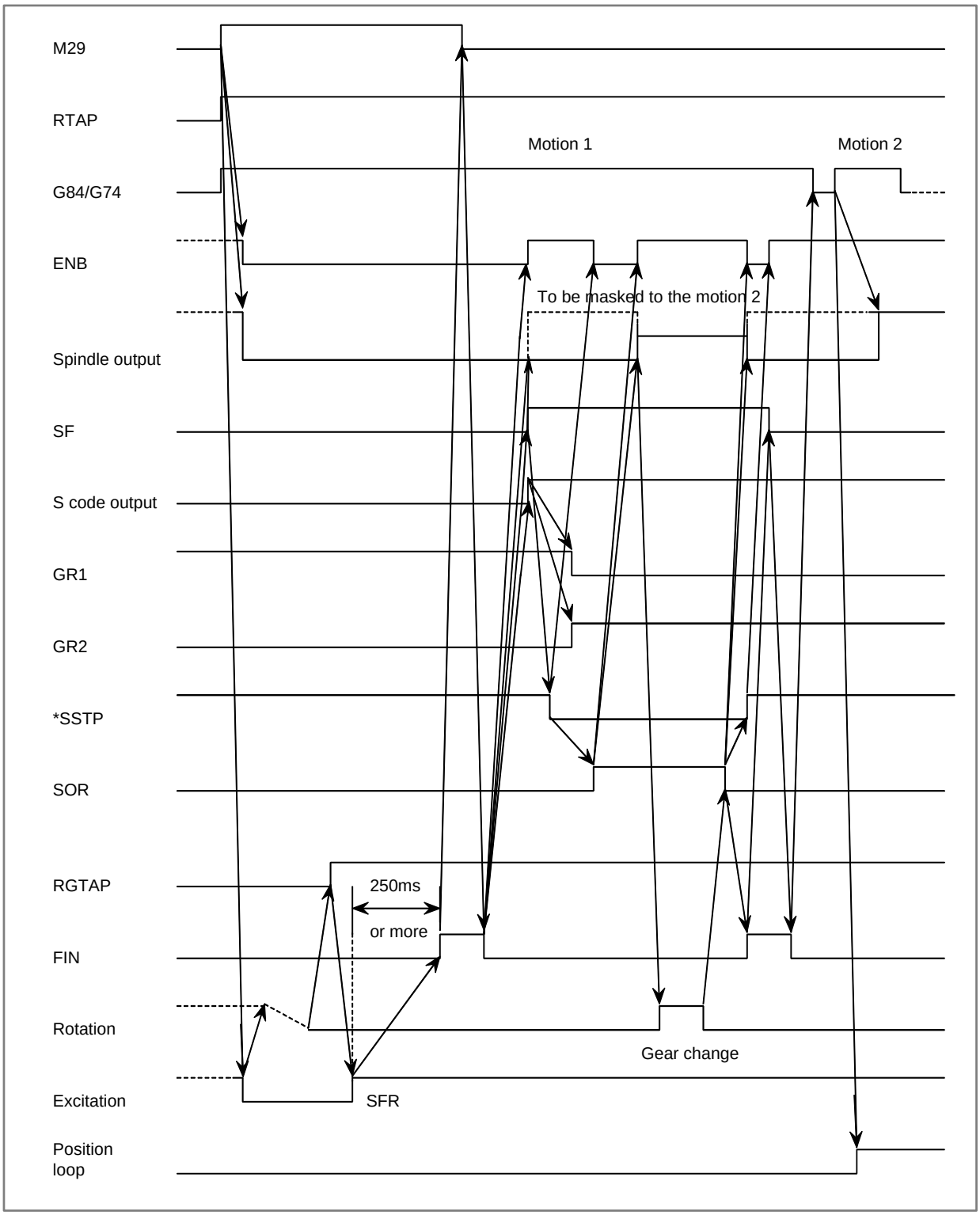


Fig. 10.13.16 (h) When gear-change is performed (middle to high gear)

NOTE

This time chart shows an example where the gear has changed from middle to high gear. The PMC selects a required gear through an S code output and selects the gear using the GR1 and GR2 signals to inform CNC of the selected gear.

10.13.16.3 Specifying G84/G74 for rigid tapping by parameters

- **M type gear selection**

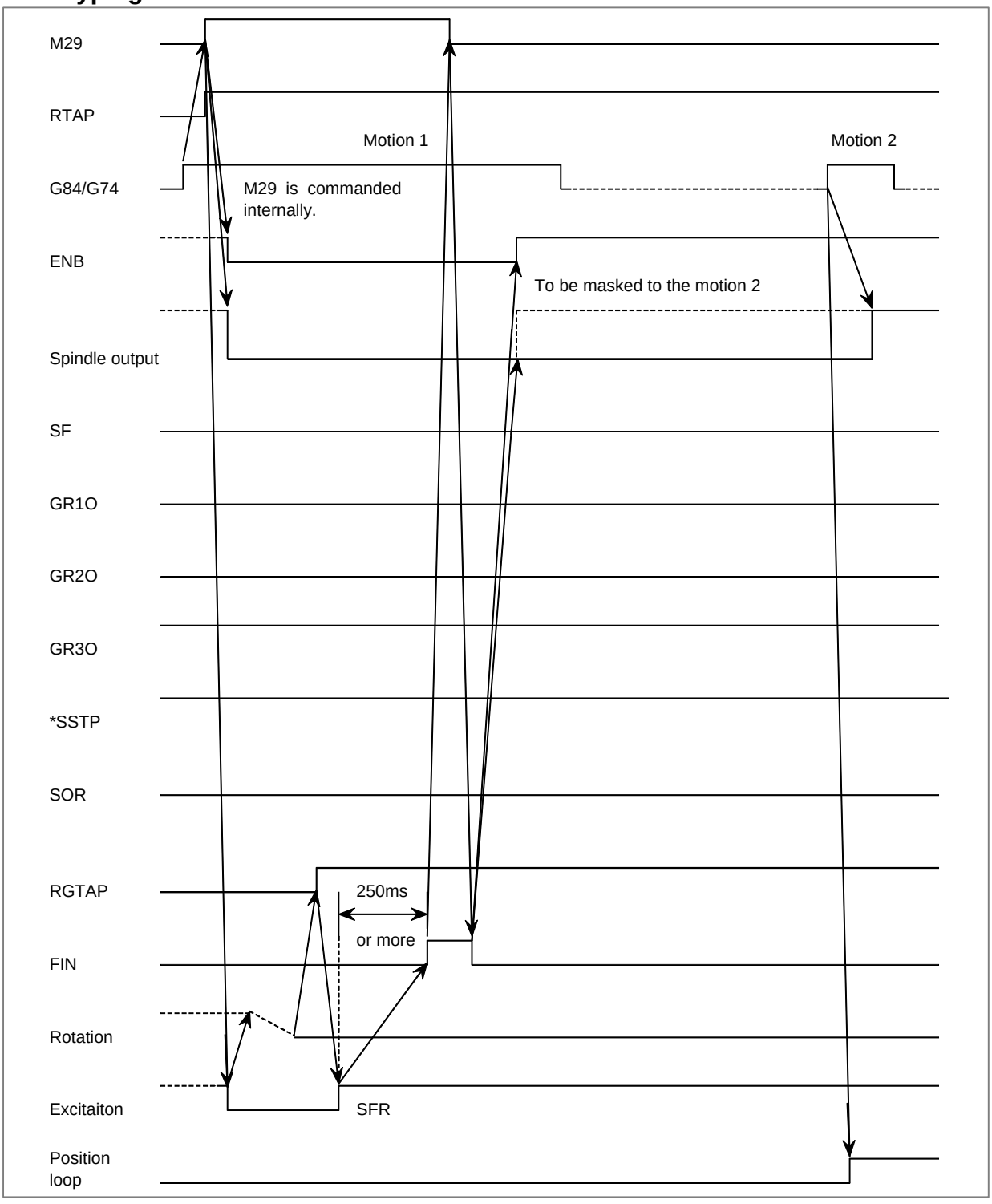


Fig. 10.13.16 (i) When gear-change is not performed

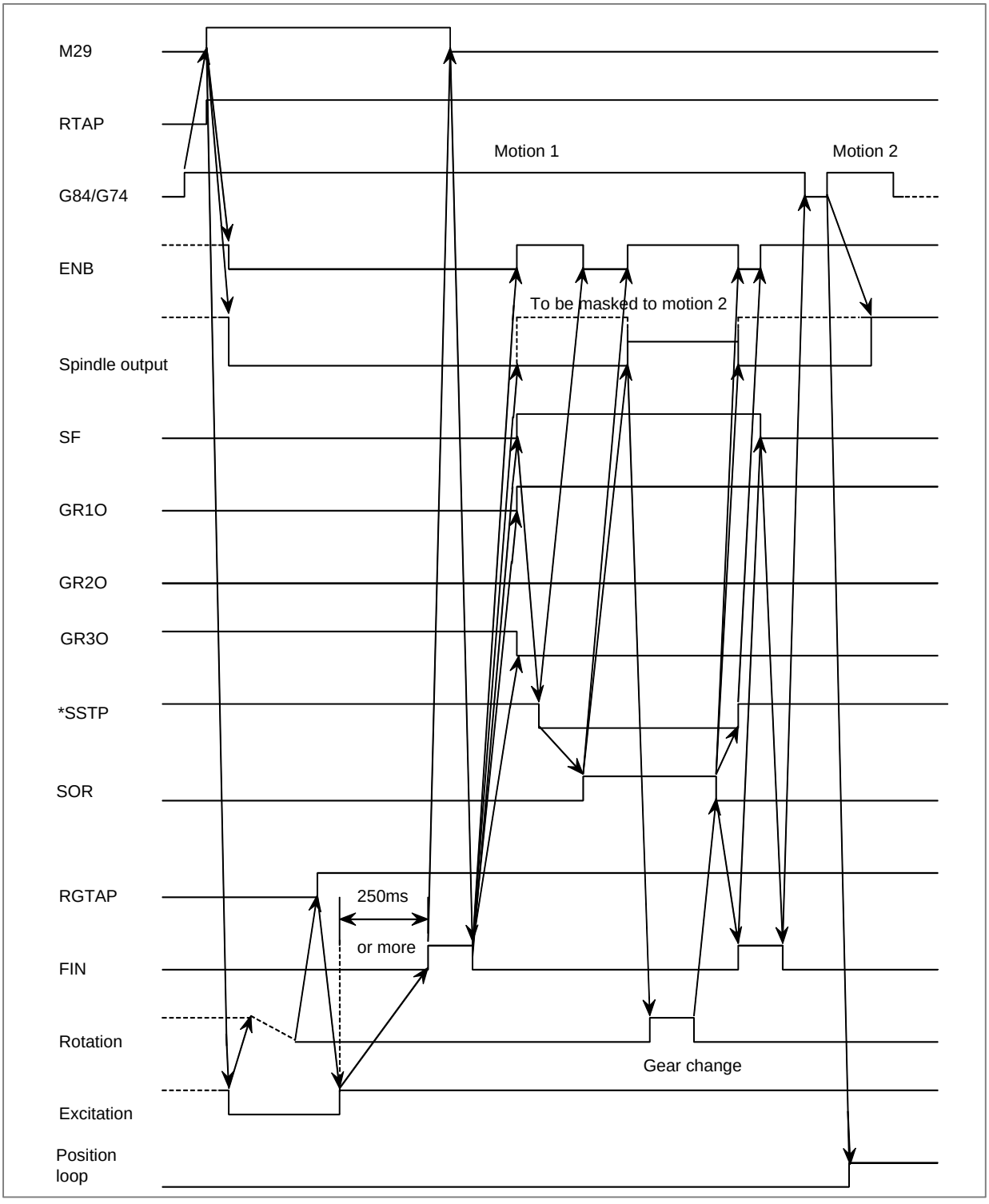


Fig. 10.13.16 (j) When gear change is performed (high to low gear)

NOTE

This time chart shows an example where the gear has shifted from high to low gear. One of the gear select signals (GR1O, GR2O, GR3O) has turned from "1" to "0", and one of the two remaining signals has turned from "0" to "1". This changes the gear.

- T type gear selection method

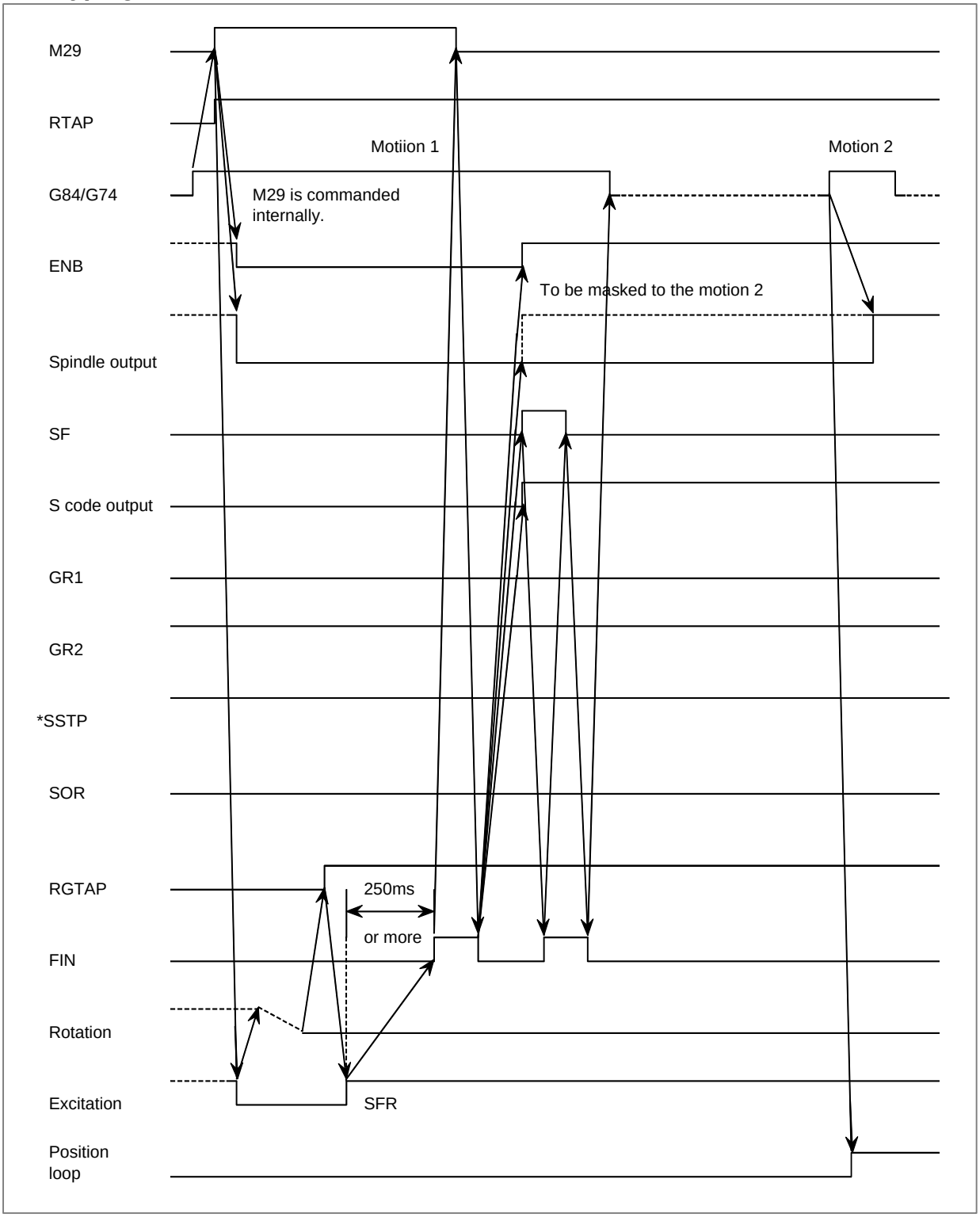


Fig. 10.13.16 (k) When gear change is not performed

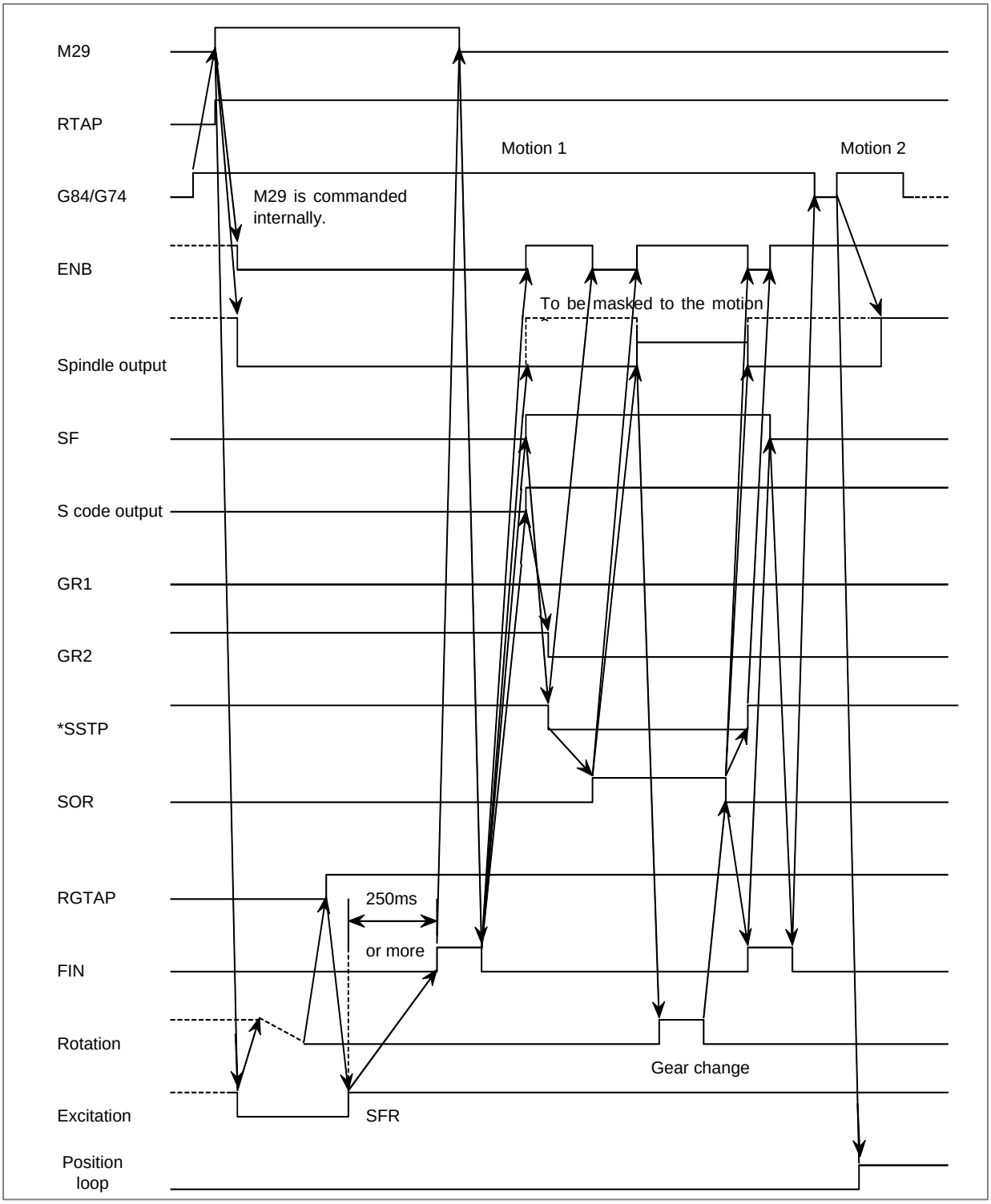


Fig. 10.13.16 (l) When gear-change is performed (high to low gear)

NOTE
This time chart shows an example where the gear has changed from high to low gear. The PMC selects a required gear through an S code output and inputs the selected gear using the GR1 and GR2 signals to inform CNC of the selected gear.

10.13.16.4 When M29 is specified before G84/G88

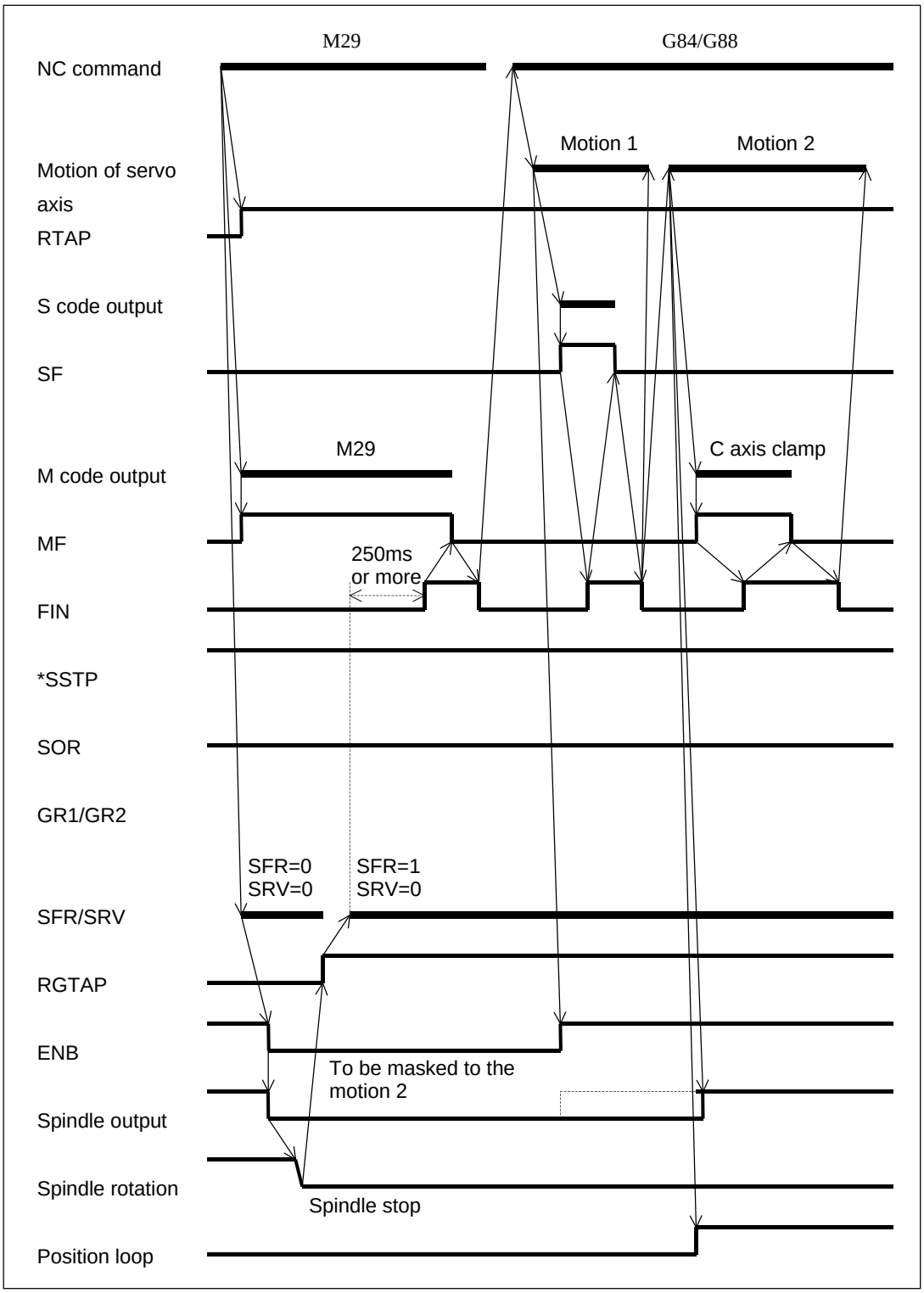


Fig. 10.13.16 (m) When gear change is not performed

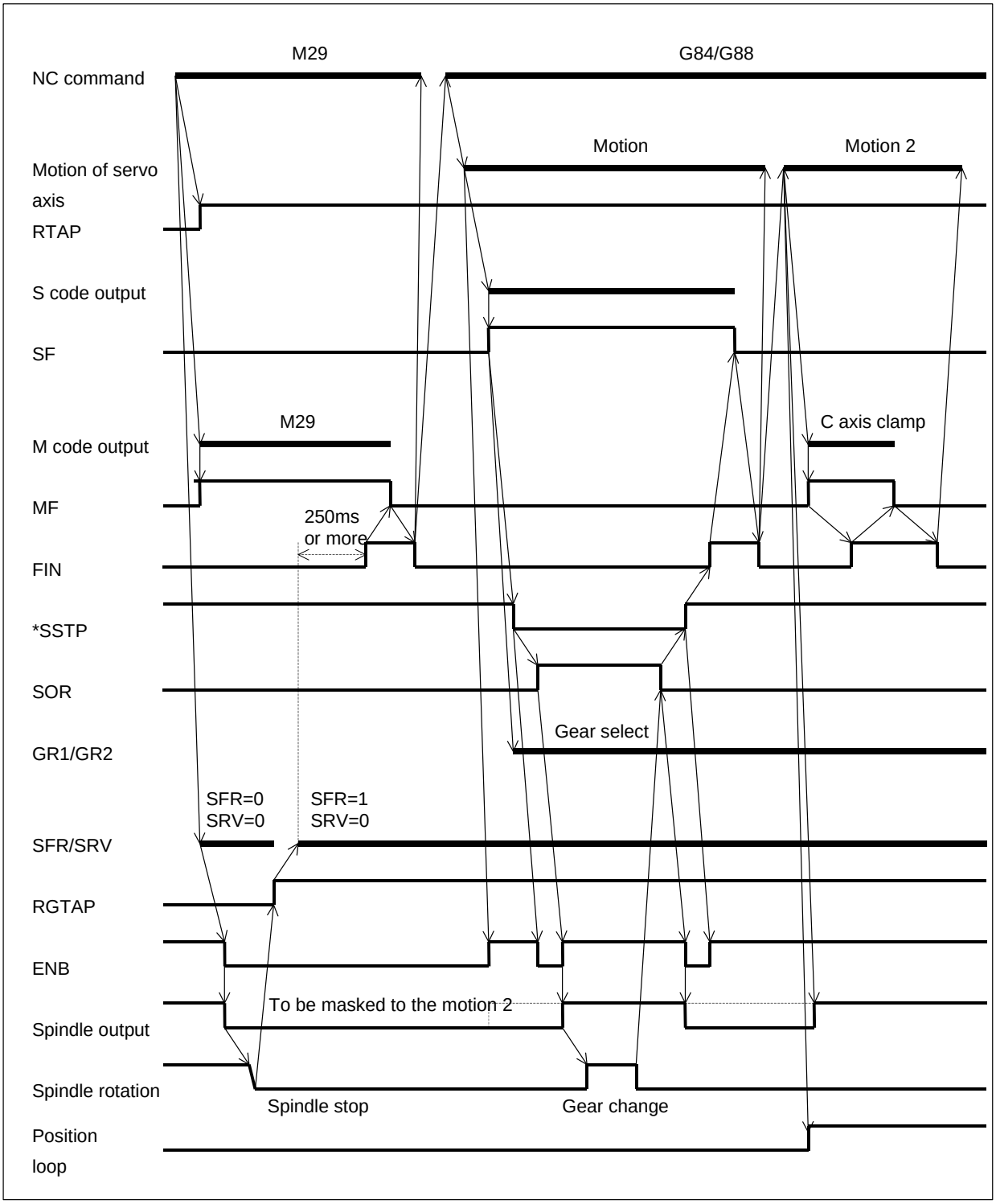


Fig. 10.13.16 (n) When gear change is performed

10.13.16.5 M29 and G84/G88 are specified in the same block

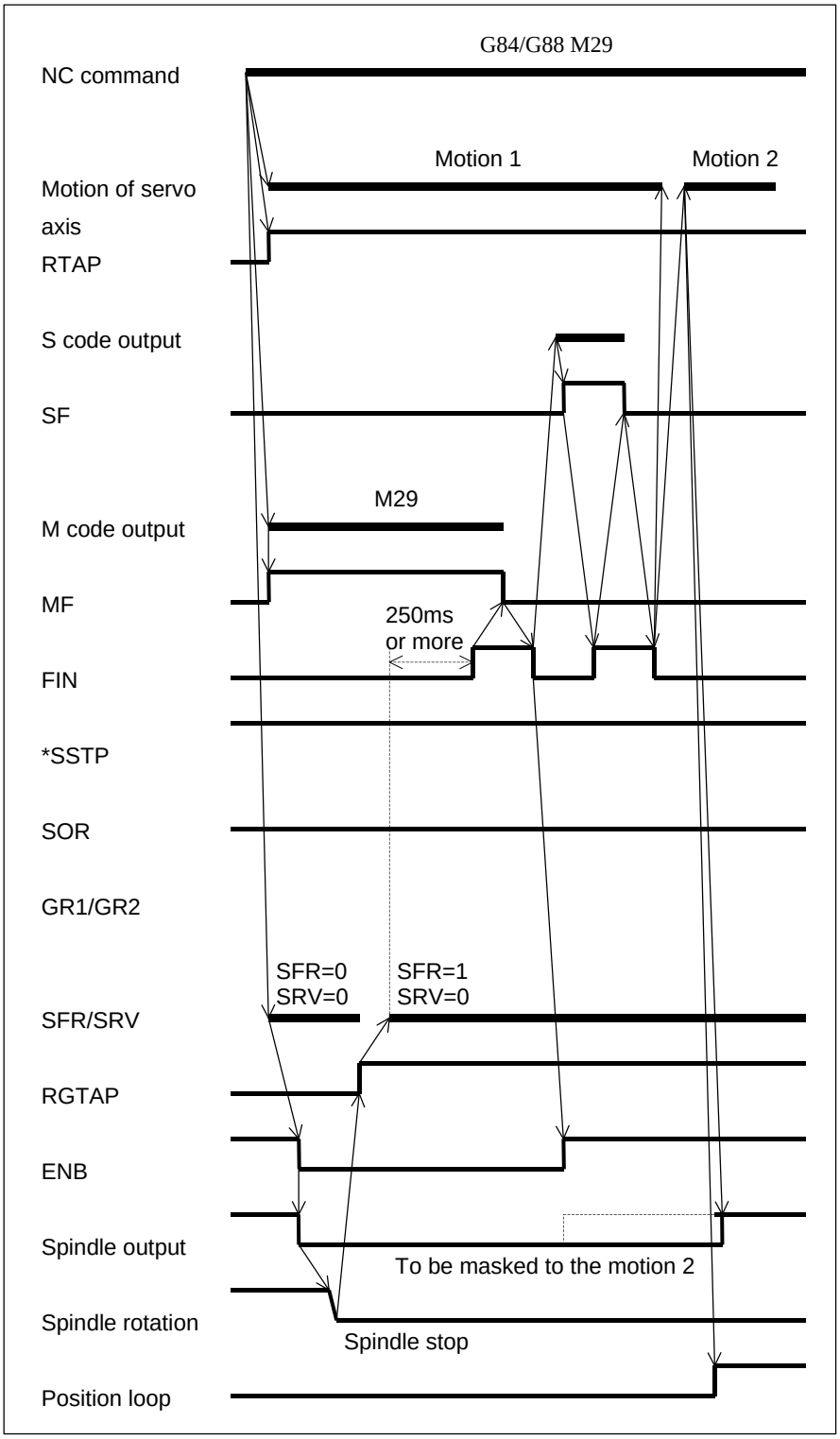


Fig. 10.13.16 (o) When gear-change is not performed

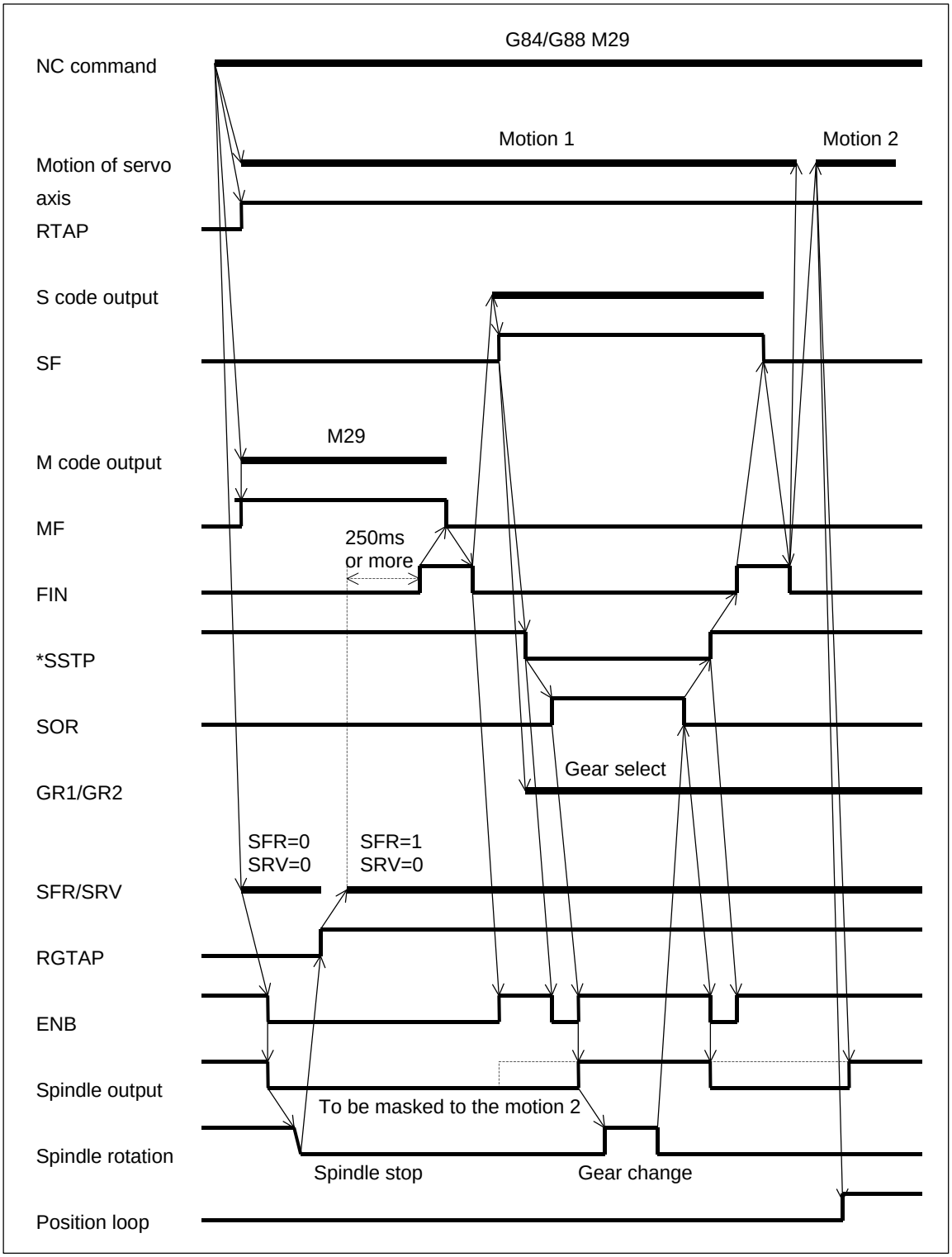


Fig. 10.13.16 (p) When gear-change is performed

10.13.16.6 Specifying G84/G88 for rigid tapping by parameters

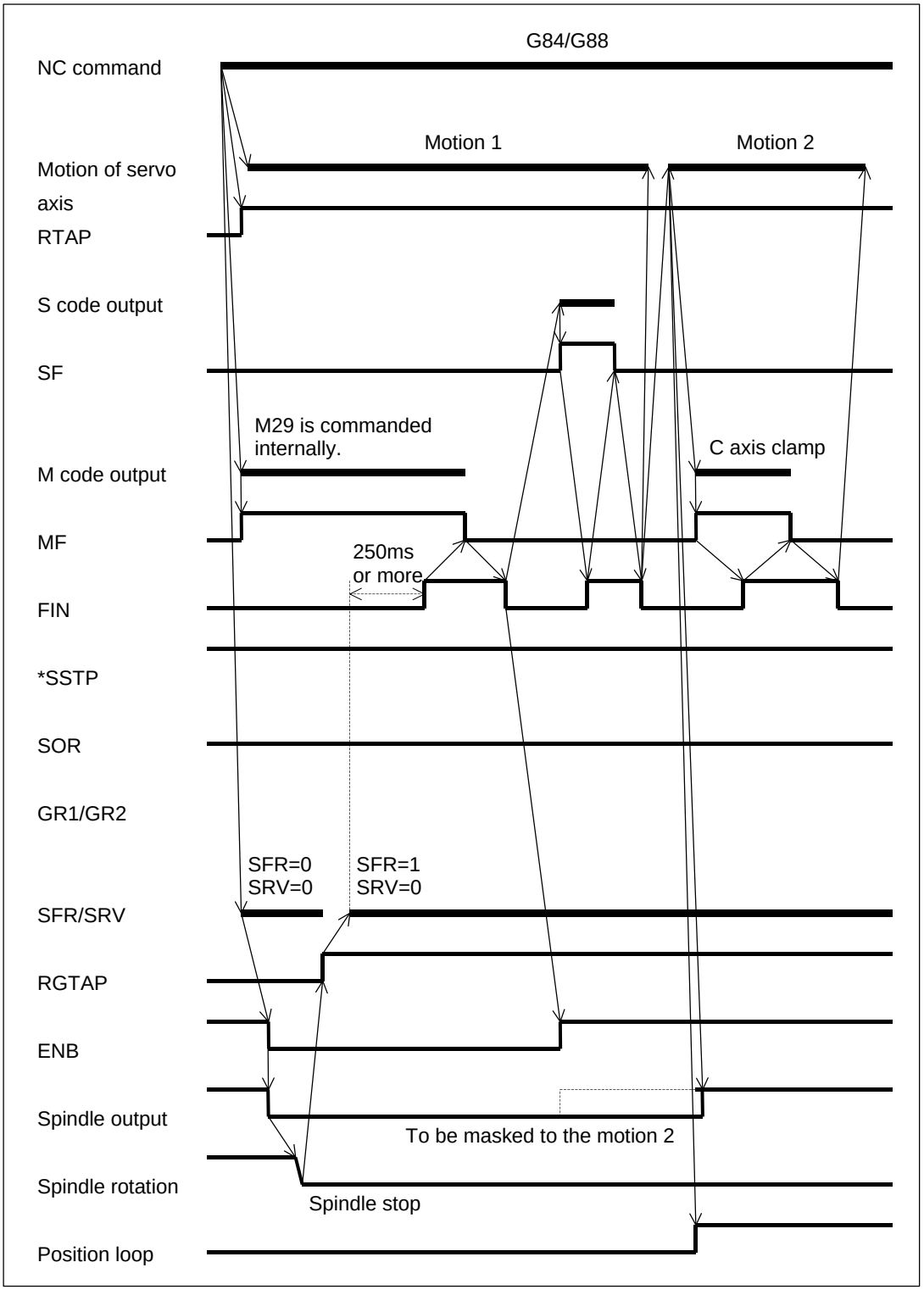


Fig. 10.13.16 (q) When gear-change is not performed

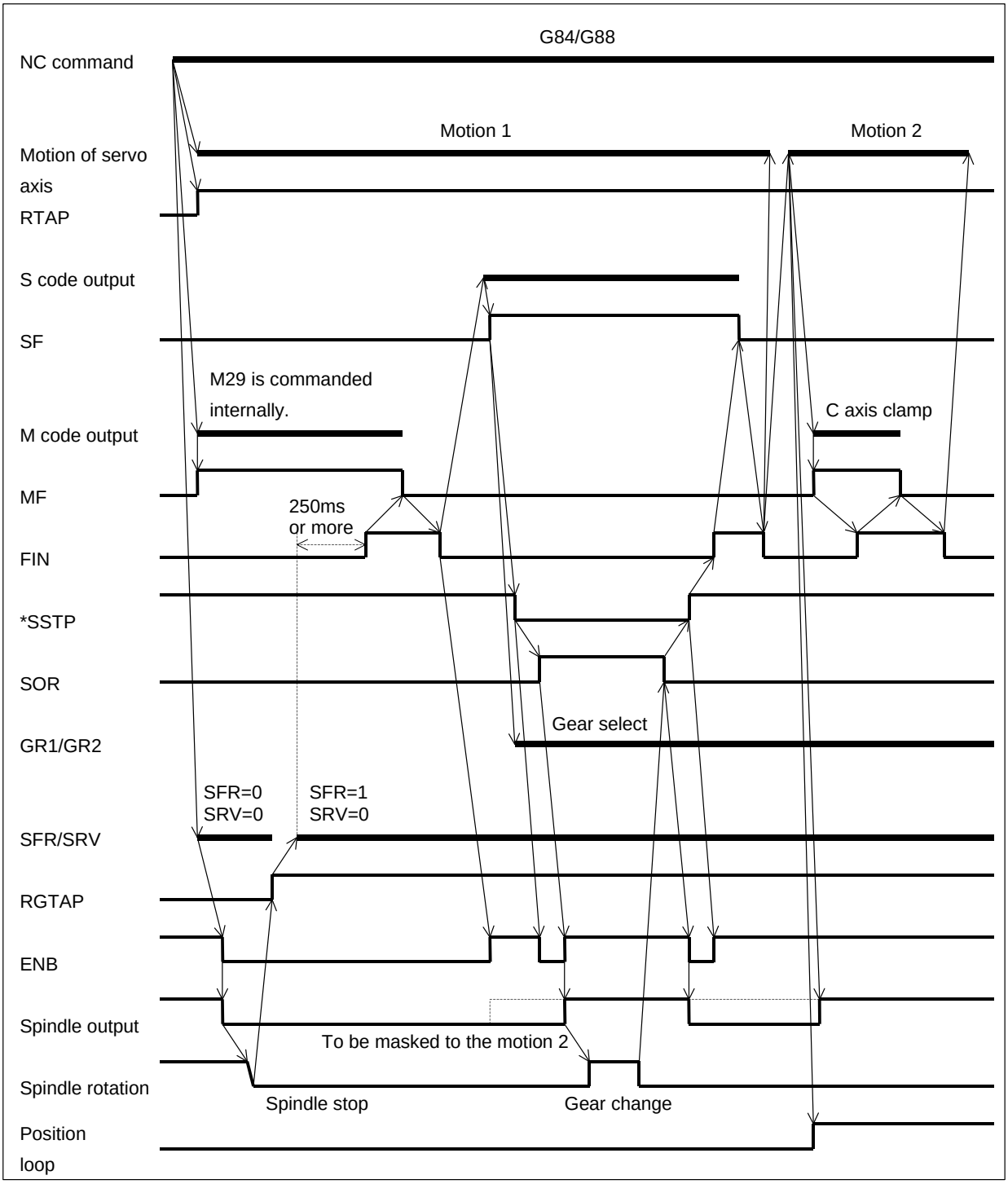
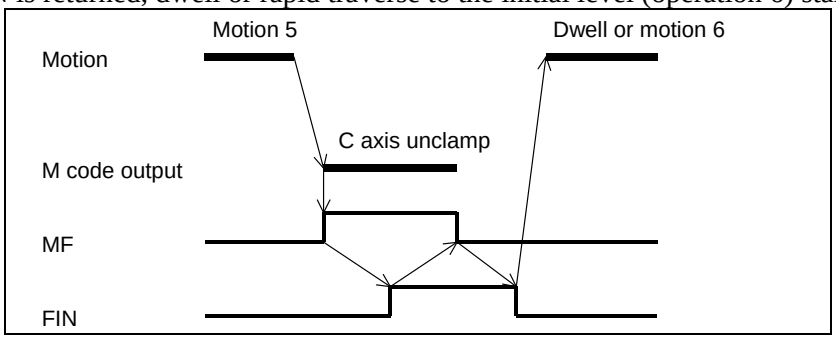


Fig. 10.13.16 (r) When gear-change is performed

10.13.16.7 Timing of the M code for unclamping (T series)

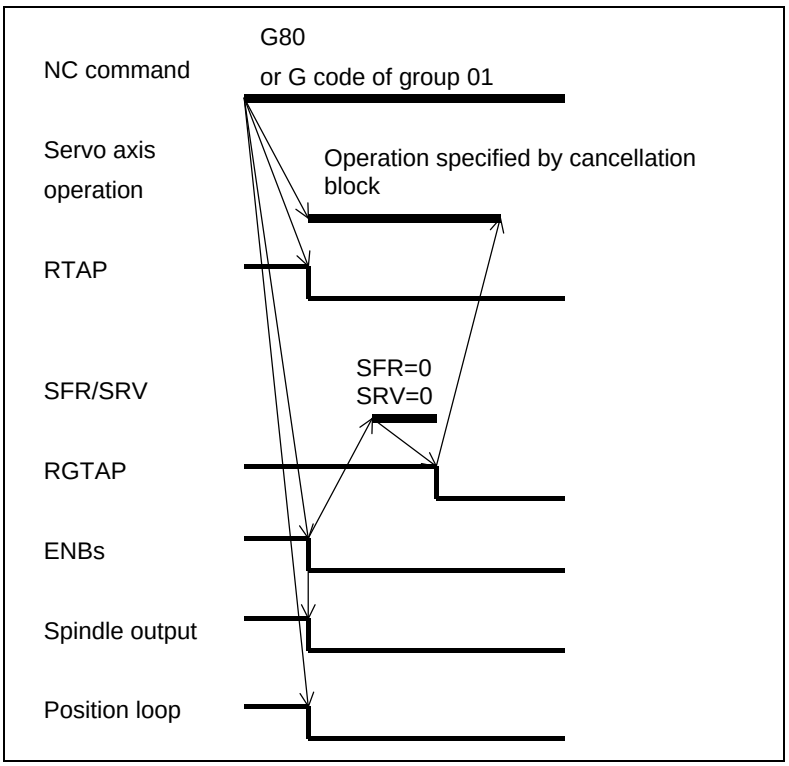
T

After extraction from the hole bottom to the R point level (operation 5), the M code for unclamping is output. When FIN is returned, dwell or rapid traverse to the initial level (operation 6) starts.



10.13.16.8 Timing to cancel rigid tapping mode

When rigid tapping is completed, the mode is canceled if a G code (such as G80, canned cycle G code, or Group 01 G code) is issued.
Spindle output has the effect of execution of S0. On the falling edge of the ENB signal (corresponding ENBs signal in multi spindle control), cancel the rigid tapping mode of the PMC (for example, by turning off spindle activation) and set the rigid tapping signal to 0. After confirming that the rigid tapping signal is set to 0, proceed to the next block.
When gear change is performed using *SSTP and SOR, the ENB signal can be either "1" or "0". Do not cancel the PMC's rigid tapping mode at the falling edge of the ENB signal under these circumstances. The position loop is also canceled.
When the CNC is reset, the PMC's rigid tapping mode must be canceled.
When CRG (parameter No. 5200#2) is "1", the system goes directly to the next block without checking that the rigid tapping signal is "0". Set CRG to "1" for systems in which the rigid tapping signal is always "1".



⚠ CAUTION

1 If rigid tapping mode is canceled by a Group 01 G code, such as G00 or G01, the block containing the G code is executed at the same time the ENB signal is turned to "0". Therefore, if the block contains an M code for controlling the spindle, an error may occur during processing in the PMC.
2 When CRG (Parameter No. 5200#2) is 1, if the next block contains an M code for controlling the spindle, an error may occur during processing in the PMC, when:
- Rigid tapping mode is canceled by issuing G80
- Rigid tapping mode is canceled by issuing a Group 01 G code , such as G00 or G01

NOTE

Rigid tapping mode is canceled as described above regardless of the gear selection method of M-type or T-type.

10.13.17 Parameter

	#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.

NOTE

1 To enable multi-spindle control, also set bit 3 (MSP) of parameter No. 8133 to 1.
2 When 2-path control is performed, this parameter is used. It must be set for a path that does not require multi-spindle control.

	#7	#6	#5	#4	#3	#2	#1	#0
3703				SPR	MPP			2P2
					MPP			

[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 2P2 When a 2-path system is used, inter-path spindle control allows:
0: Configuration where the spindle that belongs to one path only is shared between path 1 and path 2.
1: Configuration where the spindles that belong to path 1 and 2 are shared between the two paths.

#3 MPP In multi-spindle control, a spindle selection using a programmed command instead of using the signals (SWS1 to SWS2 <G027.0 to 1>) is:
0: Not made.
1: Made.

> **NOTE**
> 1 When this parameter is set to 1, set parameter No. 3781 at the same time.
> 2 When this parameter is set to 1, setting bit 6 (MPC) of parameter No. 3713 to 1 allows position coder feedback used for operations such as thread cutting and feed per revolution to also be switched automatically according to the selected spindle. (Position coder selection by address P)

#4 **SPR** Rigid tapping with spindle of another path function is:
0: Not available.
1: Available.

The "rigid tapping with spindle of another path" function can be combined with the address P-based spindle select function. This combination enables program commands (P code) to be used to select a spindle in a path other than the command issuing path for rigid tapping.

	#7	#6	#5	#4	#3	#2	#1	#0
3705		SFA		EVS	SGT			ESF
		SFA	NSF		SGT			ESF

[Input type] Parameter input
[Data type] Bit path

#0 ESF When constant surface speed control is selected (bit 0 (SSC) of parameter No. 8133 is 1) or bit 4 (GTT) of parameter No. 3706 is 1:
0: S codes and SF are output for all S commands.
1: For the T series:
S codes and SF are not output for an S command in the constant surface speed control (G96) mode and a command for maximum spindle speed clamping (G92S-- - ; (G50 for G code system A)).
For the M series:
S codes and SF are not output for an S command in the constant surface speed control (G96) mode.

> **NOTE**
> The operation of this parameter varies between the T series and M series.
> For the T series: This parameter is valid when bit 4 (EVS) of parameter No. 3705 is set to 1.
> For the M series: For an S command for maximum spindle speed clamping (G92S-- - ;), SF is not output, regardless of the setting of this parameter.

#3 SGT Gear switching method during tapping cycle (G84 and G74)
0: Method A (Same as the normal gear switching method)
1: Method B (Gears are switched during tapping cycle according to the spindle speed set in parameters 3761 and 3762).

#4 EVS S codes and SF are:
0: Not output for an S command.
1: Output for an S command.
The output of S codes and SF for an S command in constant surface speed control mode (G96), or for an S command used to specify maximum spindle speed clamping (G50S---;) depends on the setting of bit 0 (ESF) of parameter No.3705.

#5 NSF For the M series, if T type gear is selected (bit 4 (GTT) of parameter No. 3706 is 1 or constant surface speed control is enabled (bit 0 (SSC) of parameter No. 8133 is 1)), when an S code is specified:
0: SF is output.
1: SF is not output.

NOTE
This parameter does not affect S code output. For an S command for maximum spindle speed clamping (G92S-- - ;), SF is not output, regardless of the setting of this parameter.

#6 SFA The SF signal is output:
0: When gears are switched.
1: Irrespective of whether gears are switched.

	#7	#6	#5	#4	#3	#2	#1	#0
3706								
				GTT				

[Input type] Parameter input
[Data type] Bit path

#4 GTT Selection of a spindle gear selection method
0: Type M.
1: Type T.

NOTE
1 M type
The gear selection signal is not input. The CNC selects a gear based on the speed range of each gear set by a parameter beforehand according to S codes, and the selected gear is posted by outputting the gear selection signal. Moreover, the spindle speed matching the gear selected by the output gear selection signal is output.
T type
The gear selection signal is input. The spindle speed matching the gear selected by this signal is output.
2 When constant surface speed control is selected (bit 0 (SSC) of parameter No. 8133 is 1), the T type is assumed regardless of the setting of this parameter.

NOTE
3 When type T spindle gear switching is selected, the following parameters have no effect:
No.3705#2(SGB), No.3751, No.3752,
No.3705#1(GST), No.3705#3(SGT), No.3761, No.3762,
No.3705#6(SFA), No.3735, No.3736
On the other hand, parameter No. 3744 becomes usable.

3720	Number of position coder pulses

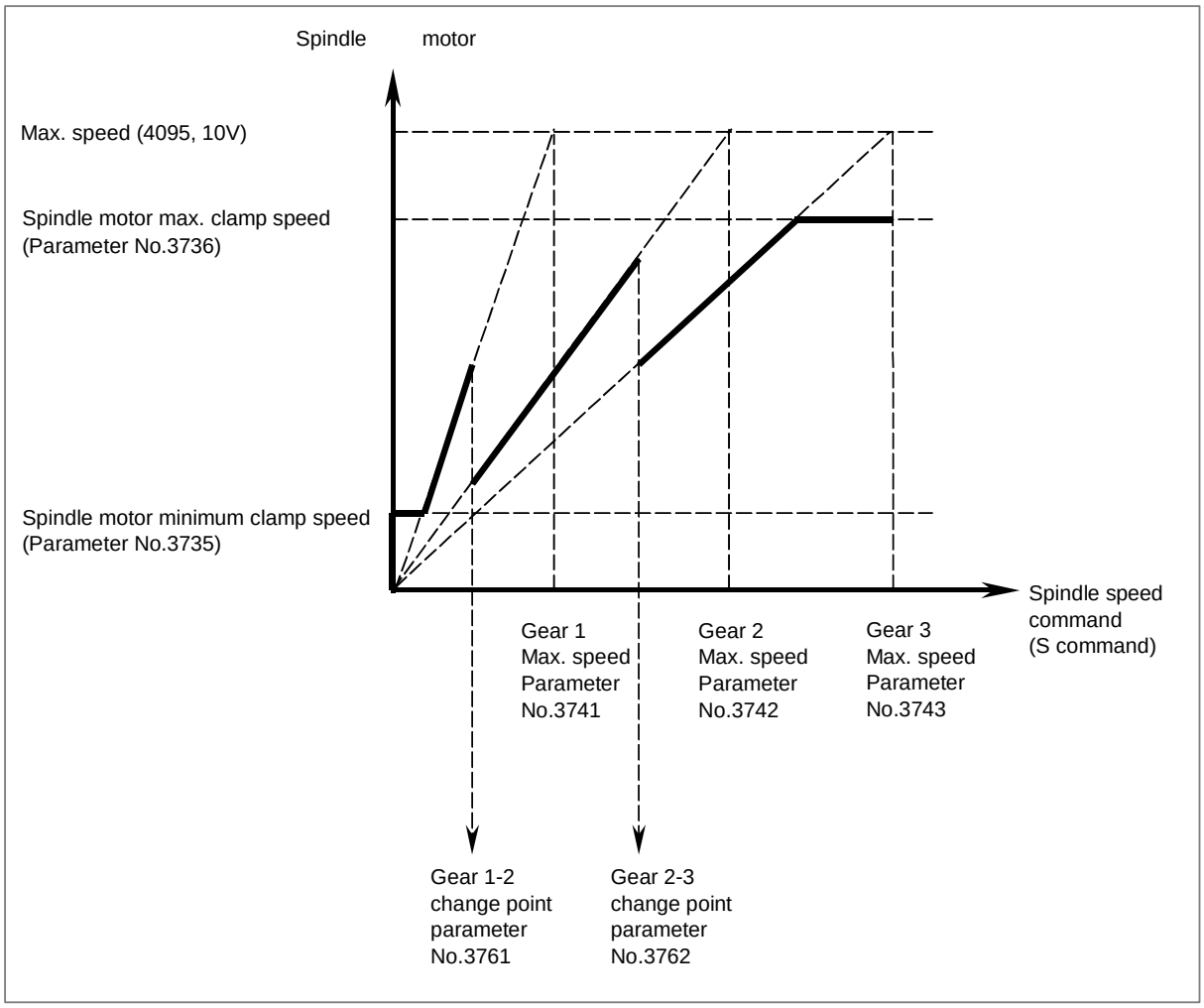
NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] Detection unit
[Valid data range] 1 to 32767
Set the number of position coder pulses.

3761	
	Spindle speed when switching from gear 1 to gear 2 during tapping

3762	
	Spindle speed when switching from gear 2 to gear 3 during tapping

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999
When method B is selected as the gear change method in the tapping cycle (when bit 3 (SGT) of parameter No. 3705 is set to 1), set the spindle speed at a change point of each gear.



	#7	#6	#5	#4	#3	#2	#1	#0
4000				RETSV				

[Data type] Bit spindle

#4 RETSV Reference position return direction in the servo mode (rigid tapping, etc.)

0: The spindle returns to the reference position counterclockwise (CCW).

1: The spindle returns to the reference position clockwise (CW).

NOTE
This parameter is valid only when a serial spindle is used.

4073	Grid shift amount in the servo mode

[Input type] Parameter input

[Data type] Word spindle

[Unit of data] 1 pulse (360°/4096)

[Valid data range] 0 to 4095

Set this data when the reference position is to be shifted in the servo mode (rigid tapping, etc.).

When plus data is specified, the reference position for the spindle is shifted CCW by the set number of pulses.

NOTE
This parameter is valid only when a serial spindle is used.

	#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 is:
0: Always the Z-axis
1: The axis selected by the program

NOTE
1 In the case of the T series, this parameter is valid only for the drilling canned cycle in the Series 10/11 format.
2 When this parameter is 1, the drilling axis determined by plane selection (G17/G18/G19) in the drilling canned cycle in the T series 10/11 format. Therefore, the Y-axis is required to specify G17/G19.

	#7	#6	#5	#4	#3	#2	#1	#0
5200	SRS	FHD	PCP	DOV	SIG	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, G01 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".

#3 SIG When gears are changed for rigid tapping, the use of SINDs is
0: Not permitted.
1: Permitted.

#4 DOV Override during extraction in rigid tapping
0: Invalidated
1: Validated (The override value is set in parameter No.5211. However, set an override value for rigid tapping return in parameter No. 5381.)

#5 PCP Address Q is specified in a tapping cycle/rigid tapping:
0: A high-speed peck tapping cycle is assumed.
1: A peck tapping cycle is assumed.

#6 FHD Feed hold and single block in rigid tapping
0: Invalidated
1: Validated

#7 SRS To select a spindle used for rigid tapping in multi-spindle control:
0: The spindle selection signals SWS1 and SWS2 are used. (These signals are used also for multi-spindle control.)
1: The rigid tapping spindle selection signals RGTSP1 and RGTSP2 are used. (These signals are provided expressly for rigid tapping.)

	#7	#6	#5	#4	#3	#2	#1	#0
5201				OV3	OVU	TDR		

[Input type] Parameter input
[Data type] Bit path

#2 TDR Cutting time constant in rigid tapping
0: Uses a same parameter during cutting and extraction (Parameter Nos. 5261 through 5264)
1: Not use a same parameter during cutting and extraction
Parameter Nos. 5261 to 5264: Time constant during cutting
Parameter Nos. 5271 to 5274: Time constant during extraction

#3 OVU The increment unit of the override parameter (No.5211) for tool rigid tapping extraction is:
0: 1%
1: 10%

#4 OV3 A spindle speed for extraction is programmed (address J), so override for extraction operation is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5202		OVE						ORI

[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 ORI When rigid tapping is started:
0: Spindle orientation is not performed.
1: Spindle orientation is performed.
This parameter can be used only for a serial spindle.
This spindle orientation performs reference position return in the serial spindle/servo mode. The stop position can be changed using the serial spindle parameter No. 4073.

#6 OVE The specification range of extraction override command (address J) by rigid tapping program specification is:
0: 100% to 200%.
1: 100% to 2000%.

NOTE
1 To enable the extraction override command (address J) by program specification, set bit 4 (OV3) of parameter No.5201 to 1.
2 When this parameter is set to 1, the operation equivalent to that of the FS0*i*-C is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
5203				OVS		RFF		
			RBL	OVS		RFF		

[Input type] Parameter input
[Data type] Bit path

#2 RFF In rigid tapping, feed forward is:
0: Disabled.
1: Enabled. (Recommended)

As the standard setting, set 1.
At the same time, set the parameter for the advanced preview feed forward coefficient for the tapping axis and the parameter for the advance preview feed forward coefficient for the spindle so that these values match.
- Advanced preview feed forward coefficient for the tapping axis: Parameter No.2092 (or parameter No.2144 if the cutting/rapid traverse feed forward function is enabled (bit 4(FFCHG) of parameter No.2214 is set to 1))
- Advanced preview feed forward coefficient for the spindle: Parameter No.4344

NOTE
This parameter is valid when a serial spindle is used.

#4 OVS In rigid tapping, override by the feedrate override select signal and cancellation of override by the override cancel signal is:
0: Disabled.
1: Enabled.
When feedrate override is enabled, extraction override is disabled.
The spindle override is clamped to 100% during rigid tapping, regardless of the setting of this parameter.

#5 RBL As acceleration/deceleration for rigid tapping cutting feed:
0: Linear acceleration/deceleration is used.
1: Bell-shaped acceleration/deceleration is used.

NOTE
The rigid tapping bell-shaped acceleration/deceleration option is required.

	#7	#6	#5	#4	#3	#2	#1	#0
5209							RIP	RTX
							RIP	

[Input type] Parameter input
[Data type] Bit path

#0 RTX In rigid tapping in a T series, the tapping axis is:
0: Selected by selecting a plane.
1: Always assumed to be the Z-axis for G84 or the X-axis for G88.

NOTE
This parameter becomes invalid when bit 1 (FCV) of parameter No.0001 is set to 1, and rigid tapping is specified using the Series10/11 format.

#1 RIP When a movement from the initial point to point R is made, the in-position check is:
0: Dependent on the setting of bit 5 (NCI) of parameter No. 1601.
1: Performed.

NOTE
This parameter is valid when bit 5 (NCI) of parameter No.1601 is set to 1 and bit 4 (IRR) of parameter No.5202 is set to 0.
If bit 5 (NCI) of parameter No.1601 is set to 0, the in-position check is performed regardless of the setting of this parameter.

5210 **Rigid tapping mode specification M code**

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 65535
This parameter sets an M code that specifies the rigid tapping mode.
The M code is judged to be 29 (M29) when "0" is set.

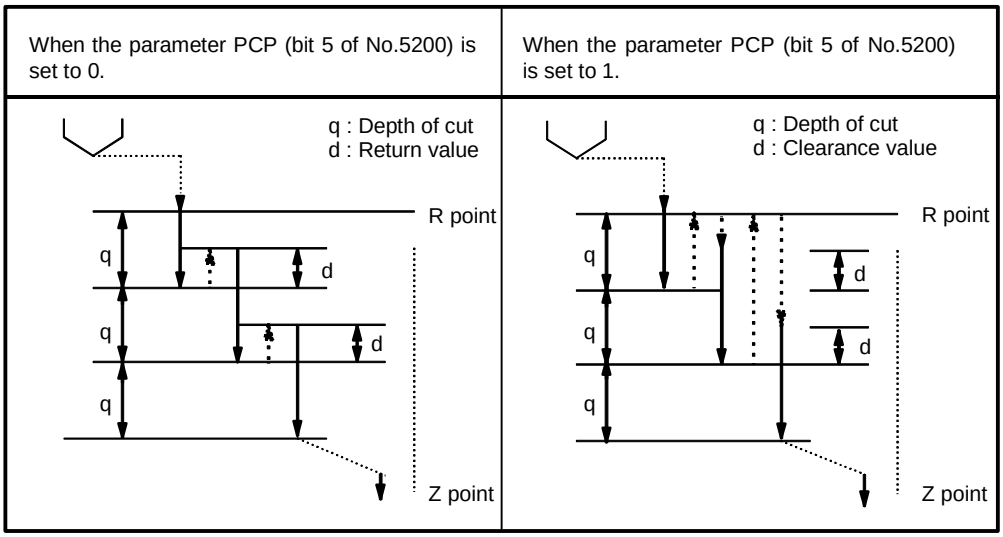
5211 **Override value during rigid tapping extraction**

[Input type] Parameter input
[Data type] Word path
[Unit of data] 1% or 10%
[Valid data range] 0 to 200
The parameter sets the override value during rigid tapping extraction.

NOTE
The override value is valid when DOV in parameter No.5200 #4 is "1". When OVU (bit 3 of parameter No.5201) is 1, the unit of set data is 10%. An override of up to 2000% can be applied to extraction.

5213 **Return in peck rigid tapping cycle**

[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 drilling 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 escape value of a high-speed peck tapping cycle or the clearance value of a peck tapping cycle.



NOTE
1 In a tapping cycle, this parameter is valid when bit 6 (PCT) of parameter No. 5104 is 1.
2 For the diameter axis, set this parameter using the diameter value.

5214	Setting of an allowable rigid tapping synchronization error range

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

This parameter sets an allowable synchronization error range in rigid tapping.
If a synchronous error range exceeding the setting of this parameter is detected, the alarm (SP0741) is issued. When 0 is set in this parameter, no synchronization error check is made.

5221	Number of gear teeth on the spindle side in rigid tapping (first gear)

5222	Number of gear teeth on the spindle side in rigid tapping (second gear)

5223	Number of gear teeth on the spindle side in rigid tapping (third gear)

5224	Number of gear teeth on the spindle side in rigid tapping (fourth gear)

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 1 to 32767

Each of these parameters is used to set the number of gear teeth on the spindle side for each gear in rigid tapping.

NOTE
When a position coder is attached to the spindle, set the same value for all of parameters No.5221 through No.5224.

5231	Number of gear teeth on the position coder side in rigid tapping (first gear)
5232	Number of gear teeth on the position coder side in rigid tapping (second gear)
5233	Number of gear teeth on the position coder side in rigid tapping (third gear)
5234	Number of gear teeth on the position coder side in rigid tapping (fourth gear)

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 1 to 32767

Each of these parameters is used to set the number of gear teeth on the position coder side for each gear in rigid tapping.

NOTE
When a position coder is attached to the spindle, set the same value for all of parameters No.5231 through No.5234.

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 will be issued. This applies to the M series.

5261	Time constant for acceleration/deceleration in rigid tapping for each gear (first gear)
5262	Time constant for acceleration/deceleration in rigid tapping for each gear (second gear)
5263	Time constant for acceleration/deceleration in rigid tapping for each gear (third gear)
5264	Time constant for acceleration/deceleration in rigid tapping for each gear (fourth gear)

[Input type] Parameter input

[Data type] Word spindle
[Unit of data] msec
[Valid data range] 0 to 4000

Each of these parameters is used to set a linear acceleration/ deceleration time constant for the spindle of each gear and the tapping axis in rigid tapping. Set the period required to reach each maximum spindle speed (parameters No.5241 to No.5244). The set time constant, multiplied by the ratio of a specified S value to a maximum spindle speed, is actually used as a time constant. For bell-shaped acceleration/deceleration, set a time constant for a linear portion.

5271	Time constant for acceleration/deceleration in rigid tapping extraction (first gear)
5272	Time constant for acceleration/deceleration in rigid tapping extraction (second gear)
5273	Time constant for acceleration/deceleration in rigid tapping extraction (third gear)
5274	Time constant for acceleration/deceleration in rigid tapping extraction (fourth gear)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] msec
[Valid data range] 0 to 4000

Each of these parameters is used to set a linear acceleration/ deceleration time constant for the spindle of each gear and tapping axis in extraction operation during rigid tapping. For bell-shaped acceleration/deceleration, set a time constant for a linear portion.
In interpolation type rigid tapping, linear/bell-shaped acceleration/ deceleration of constant acceleration time type is used. So, set a time constant directly for the spindle and tapping axis for each gear.

NOTE
These parameters are enabled when the parameter TDR (bit 2 of parameter No.5201) is set to 1.

5280	Position control loop gain for the spindle and tapping axis in rigid tapping (common to gears)
5281	Position control loop gain for the spindle and tapping axis in rigid tapping (first gear)
5282	Position control loop gain for the spindle and tapping axis in rigid tapping (second gear)
5283	Position control loop gain for the spindle and tapping axis in rigid tapping (third gear)
5284	Position control loop gain for the spindle and tapping axis in rigid tapping (fourth gear)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] 0.01/sec
[Valid data range] 1 to 9999

Each of these parameters is used to set a position control loop gain for the spindle and tapping axis in rigid tapping. These parameters significantly affect the precision of threading. Conduct cutting tests, and make adjustments to obtain an optimum value. When performing threading with an analog spindle, also adjust the loop gain multipliers (parameter Nos. 5291 to 5294).

NOTE
To use a varied loop gain on a gear-by-gear basis, set parameter No.5280 to 0, and set a loop gain for each gear in parameters No.5281 to No.5284. The specification of a loop gain on a gear-by-gear basis is disabled if parameter No.5280 is set to a value other than 0. In such a case, the value set in parameter No.5280 is used as a loop gain that is common to all the gears.

5291	Loop gain multiplier for the spindle in rigid tapping (first gear)
5292	Loop gain multiplier for the spindle in rigid tapping (second gear)
5293	Loop gain multiplier for the spindle in rigid tapping (third gear)
5294	Loop gain multiplier for the spindle in rigid tapping (fourth gear)

[Input type] Parameter input
[Data type] Word spindle
[Valid data range] 1 to 32767

Each of these parameters is used to set a loop gain multiplier for the spindle in rigid tapping each gear.
These parameters significantly affect the precision of threading. Optimize these parameters as well as the loop gains by conducting a cutting test.
Loop gain multiplier GC is obtained from the following equation:

$$GC = \frac{2048000 \times 360 \times PC \times E}{PLS \times SP \times L}$$

PLS Number of pulses output from the position coder (pulses/rev)
SP Number of gear teeth on the spindle side
PC Number of gear teeth on the position coder side
E Specified voltage (V) for turning the spindle motor at 1000 min^{-1}
L Angular displacement of the spindle (degrees) per spindle motor rotation
Example:
For the spindle motor and gear ratio given below, GC is calculated as follows:

$$GC = \frac{2048000 \times 360 \times 1 \times 2.2}{4096 \times 1 \times 360} = 1100$$

PLS = 4096 pulse/rev
SP = 1
PC = 1
E = 2.2 V
L = 360 deg

NOTE
1 On the assumption that the spindle motor used turns at 4500 min^{-1} at 10 V, 2.2 V is required to turn the spindle motor at 1000 min^{-1}
2 These parameters are used for analog spindles.

5300	Tapping axis in-position width in rigid tapping (first spindle)
5302	Tapping axis in-position width in rigid tapping (second spindle)

[Input type] Parameter input

[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] 0 to 32767
This parameter sets a tapping axis in-position width when rigid tapping is performed using the each spindle.

5301	Spindle in-position width in rigid tapping

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 32767
These parameters are used to set spindle in-position widths in rigid tapping.

NOTE
If an excessively large value is specified, the threading precision will deteriorate.

5310	Positional deviation limit imposed during tapping axis movement in rigid tapping (first spindle)

5350	Positional deviation limit imposed during tapping axis movement in rigid tapping (second spindle)

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
This parameter sets a positional deviation limit imposed during tapping axis movement in rigid tapping using the each spindle.

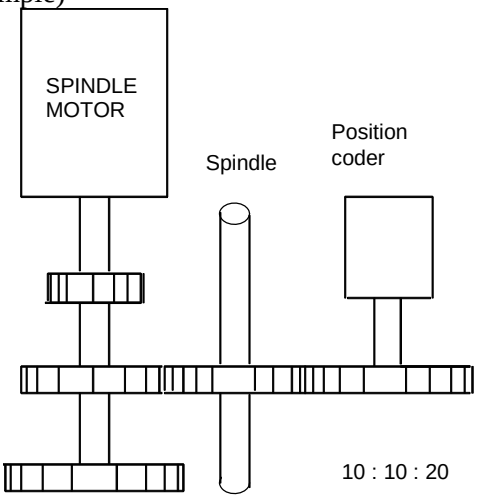
5311	Limit value of spindle positioning deviation during movement in rigid tapping

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
This parameter sets the limit value of a spindle positioning deviation during movement in rigid tapping.
Find a value to be set from the following expression:

$$\text{Setting value} = \frac{S \times PLS \times 100 \times SP \times C}{60 \times GP \times C}$$

S Maximum spindle speed in rigid tapping (min^{-1})
(Setting value of parameter Nos. 5241 and greater)
PLS Number of pulses output from the position coder (pulses/rev)
SP Number of gear teeth on the spindle side
PC Number of gear teeth on the position coder side
G Loop gain in the rigid tapping ($0.01sec^{-1}$)
(Setting value of parameter Nos. 5281 and greater)
C Coefficient 1.5

(Calculation example)



S=3600
PLS=4096
SP=10
PC=20
G=3000
C=1.5

$$\text{Setting value} = \frac{3600 \times 4096 \times 100 \times 10 \times 1.5}{60 \times 3000 \times 20} = 6144$$

5312	Positional deviation limit imposed while the tapping axis is stopped in rigid tapping (first spindle)

5352	Positional deviation limit imposed while the tapping axis is stopped in rigid tapping (second spindle)

[Input type] Parameter input
[Data type] Word axis
[Unit of data] Detection unit
[Valid data range] 0 to 32767
This parameter sets a positional deviation limit imposed while the tapping axis is stopped in rigid tapping using the each spindle.

5313	Positional deviation limit imposed while the spindle is stopped in rigid tapping

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
This parameter is used to set a positional deviation limit imposed while the spindle is stopped in rigid tapping.

5321	Spindle backlash in rigid tapping (first-stage gear)

5322	Spindle backlash in rigid tapping (second-stage gear)

5323	Spindle backlash in rigid tapping (third-stage gear)

5324	Spindle backlash in rigid tapping (fourth-stage gear)

[Input type] Parameter input

[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] -9999 to 9999
Each of these parameters is used to set a spindle backlash.

5365	Bell-shaped acceleration/deceleration time constant in rigid tapping (first-stage gear)
5366	Bell-shaped acceleration/deceleration time constant in rigid tapping (second-stage gear)
5367	Bell-shaped acceleration/deceleration time constant in rigid tapping (third-stage gear)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] msec
[Valid data range] 0 to 512
Each of these parameters is used to set a time constant for a curved portion when bell-shaped acceleration/deceleration is selected in rigid tapping. When 0 is set in this parameter, linear acceleration/ deceleration is performed.

NOTE
This parameter is enabled when the parameter RBL (bit 5 of parameter No.5203) is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
8133					MSP			SSC

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#0 SSC Constant surface speed control is:
0: Not Used.
1: Used.

#3 MSP Multi-spindle is:
0: Not Used.
1: Used.

NOTE
For the M series, multi-spindle control is optional function.

	#7	#6	#5	#4	#3	#2	#1	#0
11420								RAU

[Input type] Parameter input
[Data type] Bit path

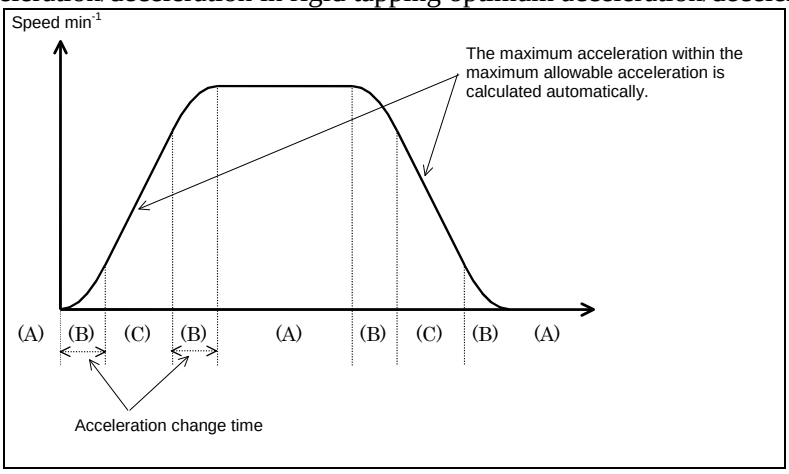
#0 RAU The rigid tapping optimum acceleration/deceleration function is:
0: Disabled.
1: Enabled.

11421	Maximum acceleration of rigid tapping optimum acceleration/deceleration (1st gear)
11422	Maximum acceleration of rigid tapping optimum acceleration/deceleration (2nd gear)
11423	Maximum acceleration of rigid tapping optimum acceleration/deceleration (3rd gear)
11424	Maximum acceleration of rigid tapping optimum acceleration/deceleration (4th gear)

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] rev/sec^2
[Valid data range] 0 to 10000.0
This parameter sets the maximum acceleration.

11425	Acceleration change time of bell-shaped acceleration/deceleration in rigid tapping optimum acceleration/deceleration (1st gear)
11426	Acceleration change time of bell-shaped acceleration/deceleration in rigid tapping optimum acceleration/deceleration (2nd gear)
11427	Acceleration change time of bell-shaped acceleration/deceleration in rigid tapping optimum acceleration/deceleration (3rd gear)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] msec
[Valid data range] 0~200
This parameter sets the acceleration change time (time required to change from constant speed state (A) to acceleration/deceleration state (C) at the acceleration calculated from rigid tapping optimum acceleration/deceleration (indicated by (B) below)) of bell-shaped acceleration/deceleration in rigid tapping optimum acceleration/deceleration.



11429	P1 spindle speed of rigid tapping optimum acceleration/deceleration (1st gear)
11430	P2 spindle speed of rigid tapping optimum acceleration/deceleration (1st gear)
11431	P3 spindle speed of rigid tapping optimum acceleration/deceleration (1st gear)
11432	P1 spindle speed of rigid tapping optimum acceleration/deceleration (2nd gear)
11433	P2 spindle speed of rigid tapping optimum acceleration/deceleration (2nd gear)
11434	P3 spindle speed of rigid tapping optimum acceleration/deceleration (2nd gear)
11435	P1 spindle speed of rigid tapping optimum acceleration/deceleration (3rd gear)
11436	P2 spindle speed of rigid tapping optimum acceleration/deceleration (3rd gear)
11437	P3 spindle speed of rigid tapping optimum acceleration/deceleration (3rd gear)
11438	P1 spindle speed of rigid tapping optimum acceleration/deceleration (4th gear)
11439	P2 spindle speed of rigid tapping optimum acceleration/deceleration (4th gear)
11440	P3 spindle speed of rigid tapping optimum acceleration/deceleration (4th gear)

[Input type] Parameter input
[Data type] Byte spindle
[Unit of data] %
[Valid data range] 0 to 100

This parameter sets the ratio of the spindle speed at acceleration setting points P1 to P3 of P0 to P4 to the maximum spindle speed (parameters Nos. 5241 to 5244). The spindle speed at P0 is 0 and the spindle speed at P4 is the maximum spindle speed. The acceleration setting point at which 0 is set is skipped.

11441	P0 allowable acceleration of rigid tapping optimum acceleration/deceleration (1st gear)
11442	P1 allowable acceleration of rigid tapping optimum acceleration/deceleration (1st gear)
11443	P2 allowable acceleration of rigid tapping optimum acceleration/deceleration (1st gear)
11444	P3 allowable acceleration of rigid tapping optimum acceleration/deceleration (1st gear)
11445	P4 allowable acceleration of rigid tapping optimum acceleration/deceleration (1st gear)
11446	P0 allowable acceleration of rigid tapping optimum acceleration/deceleration (2nd gear)
11447	P1 allowable acceleration of rigid tapping optimum acceleration/deceleration (2nd gear)
11448	P2 allowable acceleration of rigid tapping optimum acceleration/deceleration (2nd gear)
11449	P3 allowable acceleration of rigid tapping optimum acceleration/deceleration (2nd gear)

11450	P4 allowable acceleration of rigid tapping optimum acceleration/deceleration (2nd gear)
11451	P0 allowable acceleration of rigid tapping optimum acceleration/deceleration (3rd gear)
11452	P1 allowable acceleration of rigid tapping optimum acceleration/deceleration (3rd gear)
11453	P2 allowable acceleration of rigid tapping optimum acceleration/deceleration (3rd gear)
11454	P3 allowable acceleration of rigid tapping optimum acceleration/deceleration (3rd gear)
11455	P4 allowable acceleration of rigid tapping optimum acceleration/deceleration (3rd gear)
11456	P0 allowable acceleration of rigid tapping optimum acceleration/deceleration (4th gear)
11457	P1 allowable acceleration of rigid tapping optimum acceleration/deceleration (4th gear)
11458	P2 allowable acceleration of rigid tapping optimum acceleration/deceleration (4th gear)
11459	P3 allowable acceleration of rigid tapping optimum acceleration/deceleration (4th gear)
11460	P4 allowable acceleration of rigid tapping optimum acceleration/deceleration (4th gear)

[Input type] Parameter input
[Data type] Byte spindle
[Unit of data] %
[Valid data range] 0 to 100

This parameter sets the ratio of the allowable acceleration at acceleration setting points P0 to P4 to the maximum acceleration (parameters Nos. 11421 to 11424). The rate of the acceleration setting point at which 0 is set is assumed to be 100%.

11461	P0 allowable deceleration of rigid tapping optimum acceleration/deceleration (1st gear)
11462	P1 allowable deceleration of rigid tapping optimum acceleration/deceleration (1st gear)
11463	P2 allowable deceleration of rigid tapping optimum acceleration/deceleration (1st gear)
11464	P3 allowable deceleration of rigid tapping optimum acceleration/deceleration (1st gear)
11465	P4 allowable deceleration of rigid tapping optimum acceleration/deceleration (1st gear)
11466	P0 allowable deceleration of rigid tapping optimum acceleration/deceleration (2nd gear)
11467	P1 allowable deceleration of rigid tapping optimum acceleration/deceleration (2nd gear)
11468	P2 allowable deceleration of rigid tapping optimum acceleration/deceleration (2nd gear)
11469	P3 allowable deceleration of rigid tapping optimum acceleration/deceleration (2nd gear)

11470	P4 allowable deceleration of rigid tapping optimum acceleration/deceleration (2nd gear)
11471	P0 allowable deceleration of rigid tapping optimum acceleration/deceleration (3rd gear)
11472	P1 allowable deceleration of rigid tapping optimum acceleration/deceleration (3rd gear)
11473	P2 allowable deceleration of rigid tapping optimum acceleration/deceleration (3rd gear)
11474	P3 allowable deceleration of rigid tapping optimum acceleration/deceleration (3rd gear)
11475	P4 allowable deceleration of rigid tapping optimum acceleration/deceleration (3rd gear)
11476	P0 allowable deceleration of rigid tapping optimum acceleration/deceleration (4th gear)
11477	P1 allowable deceleration of rigid tapping optimum acceleration/deceleration (4th gear)
11478	P2 allowable deceleration of rigid tapping optimum acceleration/deceleration (4th gear)
11479	P3 allowable deceleration of rigid tapping optimum acceleration/deceleration (4th gear)
11480	P4 allowable deceleration of rigid tapping optimum acceleration/deceleration (4th gear)

[Input type] Parameter input
[Data type] Byte spindle
[Unit of data] %
[Valid data range] 0 to 100

This parameter sets the ratio of the allowable deceleration at acceleration setting points P0 to P4 to the maximum acceleration (parameters Nos. 11421 to 11424). The rate of the acceleration setting point at which 0 is set is assumed to be 100%.

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)

In the optimum acceleration/deceleration for rigid tapping, this parameter is effective. Set the parameter of the drilling axis.

Alarm and message

Number	Message	Description
PS0200	ILLEGAL S CODE COMMAND	In the rigid tap, an S value was out of range or was not specified. The parameter (Nos. 5241 to 5243) setting is an S value which can be specified for the rigid tap.
PS0201	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 tapping command is specified, this alarm is generated. This is because cutting is not possible by the lead specified by the program.

Number	Message	Description
PS0202	POSITION LSI OVERFLOW	In the rigid tap, spindle distribution value is too large. (System error)
PS0203	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.
PS0204	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.
PS0205	RIGID MODE DI SIGNAL OFF	(1) Although a rigid M code (M29) is specified in rigid tapping, the rigid mode DI signal (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. (2) In rigid tapping, no spindle has selected commands in the path of interest. Alternatively, spindles in two or more paths have selected commands in the same path. Check the spindle command select signal. (3) A spindle specified for rigid tapping is already being used in another path. (4) In a machine provided with the multi-spindle function, no spindle has been selected for attempted rigid tapping.
PS0206	CAN NOT CHANGE PLANE (RIGID TAP)	Plane changeover was instructed in the rigid mode. Modify the program.
PS0207	RIGID DATA MISMATCH	The specified distance was too short or too long in rigid tapping.
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.(Normally, this message is output when the setting of parameter No. 1828 is exceeded.)
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.
SP0742	RIGID TAP ALARM : LSI OVERFLOW	An LSI overflow has occurred for the spindle during rigid tapping.
SP0752	SPINDLE MODE CHANGE ERROR	The spindle is in a mode (such as Cs contour control or spindle synchronous control) other than the spindle control mode. When the "rigid tapping with spindle of another path" function is used, the spindle to be used for rigid tapping must be switched to the spindle control mode before the start of rigid tapping.
DS1711	ILLEGAL ACC PARAMETER (RIGID TAPPING OPTIMUM ACC/DEC)	The permissible acceleration parameter for rigid tapping optimum acceleration/deceleration is illegal. This alarm is caused by one of the following: <1> The ratio of the deceleration rate to the acceleration rate is less than 1/3. <2> Time until the speed is decelerated to 0 exceeds the maximum value. <3> The maximum acceleration rate (parameter No. 11421 to 11424) is set to 0.

10.13.18 Notes

- Notes on spindles

Caution

⚠ CAUTION

For the analog spindle, adjust the drift compensation value precisely. Otherwise, the tool may stop, waiting to enter the in-position width in the following cases:

- At the start of tapping
- At the end of tapping
- At the hole bottom

Note

NOTE

1 A spindle pitch error is not compensated for in rigid tapping mode.

2 The limits to the number of pulses assigned to each spindle are as follows (displayed with diagnosis display No. 451):

- Serial spindle: 32767 pulses every 8 msec
- Analog spindle: 4095 pulses every 8 msec

These values vary with the position coder gear ratio setting and rigid tapping specification. If a value greater than the maximum allowable number is specified, an alarm PS0202 is issued.

- Other cautions

⚠ CAUTION

1 When the spindle orientation function is to be used at the same time
The spindle orientation function positions the spindle by using sensors and the PMC, without being directly controlled by the CNC.
The CNC has no direct control over this processing, instead following the specifications of the spindle orientation function being used.

2 When the spindle positioning function (T series) is to be used at the same time
When the spindle positioning function (T series) is to be used together with rigid tapping, rigid tapping mode must not be specified in spindle indexing mode, and spindle indexing mode must not be specified in rigid tapping mode. (Spindle positioning and rigid tapping cannot be performed simultaneously for a single spindle.)
This is not applicable when multi spindle control is used together and a spindle used for spindle position indexing is different from a spindle used for rigid tapping.

⚠ CAUTION

3 When the spindle positioning function is to be used at the same time
When the Cs contouring control function for the serial spindle is used together with the rigid tapping function, the same motor is used for spindle rotation control, Cs contouring control, and rigid tapping modes. The following points must be noted:
(1) Whether to enter Cs contouring control mode or spindle rotation control mode is selected by the CON (Cs contouring control switch signal) signal; however, the system can enter rigid tapping mode regardless of the state of the CON signal. When the rigid tapping mode is canceled the system enters spindle rotation control mode or Cs contouring control mode according to the state of the CON signal.
(2) Since the system can change to rigid tapping mode directly from the Cs contouring control mode, use of the Cs contouring control function enables the tapping tool to be positioned before rigid tapping begins. Accurate positioning is not guaranteed. If the rigid tapping cycle executes gear change or output range changing, positioning is valid.
(3) Although the system can change to rigid tapping mode directly from Cs contouring control mode, positions designated in Cs contouring control mode are not preserved if rigid tapping mode is canceled by G80. When the system is changed to rigid tapping mode from Cs contouring control mode, then returns to the Cs contouring control mode, G00 or G28 must be issued to position the tapping tool.
(4) In systems with the serial spindle Cs contouring control function, the spindle motor is in a state called servo mode when it is operating in rigid tapping mode. In servo mode, it can accept jogging and manual handling feed. To prevent this, inhibit jogging and manual handling feed of the Cs contouring axis in the PMC logic during rigid tapping.
(5) The servo-off signal for the Cs contour control axis is valid also for the spindle during rigid tapping. It should be masked on the PMC side as required.
(6) When multi spindle control is used together and a spindle used for Cs contour control is different from a spindle used for rigid tapping, rigid tapping can be specified with one spindle while the other spindle is placed in the Cs contour control mode.

4 Program restart
The program cannot be restarted in a block placed in the rigid tapping mode. (The program cannot be restarted in blocks between an M29 command and G80 command.)

5 Specification of multiple M codes in one block
Specify an M code for specifying the rigid tapping mode as an independent M code. (Multiple M codes must not be specified in one block.)

- Position control loop gain switching and serial spindle parameters

In rigid tapping, the loop gain of the drilling axis is switched so that the loop gains for position control of the drilling axis and spindle match each other.
This switching processing is specified by parameter Nos. 5281 to 5284.
The spindle loop gain depends on the setting of serial spindle parameters and the input gear signals (CTH2, CTH1).
So, in order to perform rigid tapping with a serial spindle, the position control loop gain of the spindle must be set in the parameters for the serial spindle used for rigid tapping.

When multi spindle control is used, a spindle other than the first spindle can be used for rigid tapping. So, set the parameters below for the serial spindle used for rigid tapping.
The parameters indicated below are the major serial spindle parameters required for the setting and adjustment needed to use a serial spindle.
For details of the serial spindle parameters, refer to the "FANUC AC SPINDLE MOTOR αi series DESCRIPTIONS (B-65272E)" or "FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)".

4044	**Proportional gain of the velocity loop in servo mode (gear 1, gear 2)**
4045	**Proportional gain of the velocity loop in servo mode (gear 3, gear 4)**

[Unit of data]
[Valid data range] 0 to 32767
Set a proportional gain for the velocity loop in a servo mode (such as rigid tapping mode).

4052	**Integral gain of the velocity loop in the servo mode (gear 1, gear 2)**
4053	**Integral gain of the velocity loop in the servo mode(gear 3, gear 4)**

[Unit of data]
[Valid data range] 0 to 32767
Set an integral gain of the velocity loop in a servo mode (such as rigid tapping mode).

4065	**Position gain in the servo mode (HIGH)**
4066	**Position gain in the servo mode (MEDIUM HIGH)**
4067	**Position gain in the servo mode (MEDIUM LOW)**
4068	**Position gain in the servo mode (LOW)**

[Unit of data] $0.01sec^{-1}$
[Valid data range] 0 to 65535
Set a servo loop gain in a servo mode (such as rigid tapping mode).

> ⚠ **CAUTION**
> Set a loop gain for spindle position control in rigid tapping using a serial spindle. In these parameters, basically, set the same values as those set in parameter Nos. 5281 to 5284 (loop gains for position control of the tapping axis).
> Which serial spindle parameter (i.e., loop gain) is actually used to operate the spindle depends on the serial spindle clutch/gear selection signals CTH1 and CTH2. Accordingly, which parameter is to be used must be determined by considering the gear switching and PMC software.

The table below indicates the relationship between the spindle gear selection signals and selected gear numbers.

CTH1	CTH2	Gear selected	Parameter No. to be used		
0	0	HIGH	4065	4044	4052
0	1	MIDEUM HIGH	4066		
1	0	MIDEUM LOW	4067	4045	4053
1	1	LOW	4068		

Reference item

Manual name	Item name
OPERATOR'S MANUAL (B- 64304EN)	Rigid tapping
CONNECTION MANUAL (FUNCTION) (This manual)	Spindle speed control
	Multi spindle
FANUC SERVO AMPLIFIER αi series DESCRIPTIONS (B-65282EN)	Rigid tapping
FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)	Rigid tapping

10.14 SPINDLE SYNCHRONOUS CONTROL

Overview

This function can exercise synchronous control on spindles. This function can also control the rotation phase of spindles, so that not only a round rod but also a non-standard workpiece can be grasped by either of two spindles.
This function enables the synchronous control. It also enables the control of the rotation phase of a spindle, allowing non-standard workpieces as well as rods to be held by either of the two spindles.

A combination of a master spindle and slave spindle in spindle synchronous control can be selected arbitrarily from spindles.

This function can be used with serial spindles.

NOTE
To use spindle synchronous control, set bit 4 (SYC) of parameter No. 8133 to 1.

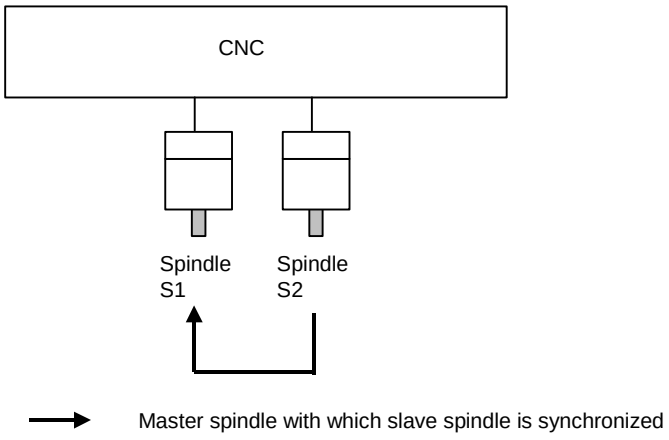
Explanation

- Synchronous-spindle configuration (when the one path system is used)

In spindle synchronous control, the spindle to which an S command is issued is called the master spindle. A spindle which ignores any S command that is issued for it, instead rotating synchronously with the master spindle, is called the slave spindle.

When the parameter SSS (No. 3704#4) is se to 0
The configuration of synchronized spindles is as follows:

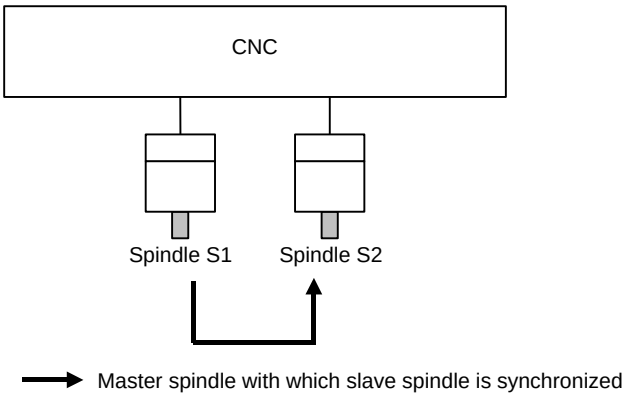
Master spindle	Slave spindle.
First spindle	Second spindle



When the parameter SSS (No. 3704#4) is se to 1
Which slave spindle is synchronized with which master spindle is set using parameter No. 4831.

Master spindle	Slave spindle.
Arbitrary spindle	Arbitrary spindle

NOTE
When an S command is to be specified with the second spindle set as the master spindle, multi-spindle control is required. (When spindle output control with the PMC (signal SINDx) is used, multi-spindle control is unnecessary.)

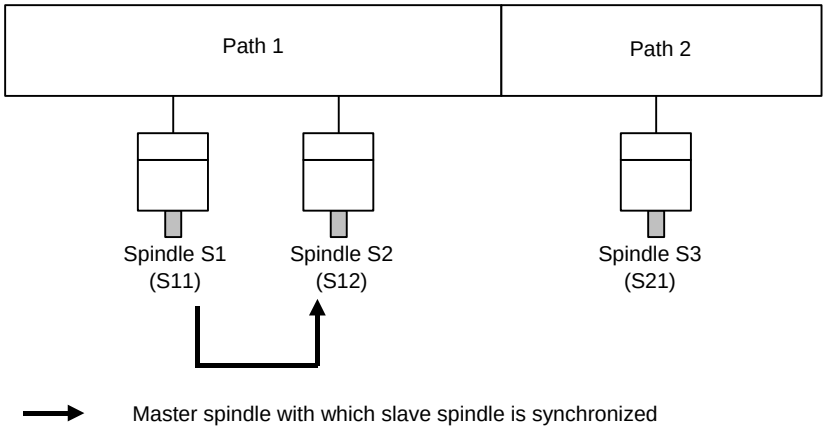


T

- Configuration of synchronized spindles (in a 2-path system)

When the parameter SSS (No. 3704#4) is se to 0
The configuration of synchronized spindles is as follows:

Master spindle	Slave spindle
First spindle of each path	Second spindle of each path



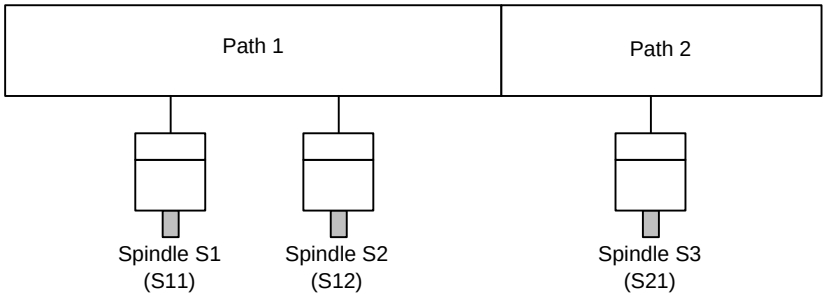
When the parameter SSS (No. 3704#4) is se to 1
Spindle synchronization can be performed by combining arbitrary spindles.
Which slave spindle is synchronized with which master spindle is set using parameter No. 4831/4832.

Master spindle	Slave spindle.
Arbitrary spindle of each path	Arbitrary spindle of each path

Set a master spindle for each slave in parameter No. 4831.
As a master spindle, set a spindle number in the same path as for a slave spindle.

By setting a logic spindle number common to the system in parameter No. 4832, such spindle synchronization that an arbitrary spindle belonging to a different path is used as a master spindle can be exercised.
The spindle number of a master spindle to be set is a logic spindle number common to the system. In this case, set 0 in parameter No. 4831 for all.
For the path to which a slave spindle belongs and the path to which a master spindle belongs, bit 4 (SSS) of parameter No. 3704 needs to be set to 1 (to enable spindle synchronization using arbitrary axes).
Example of parameter setting)
Spindle configuration of each path
Three spindles (two spindles of path 1 + one spindles of path 2)

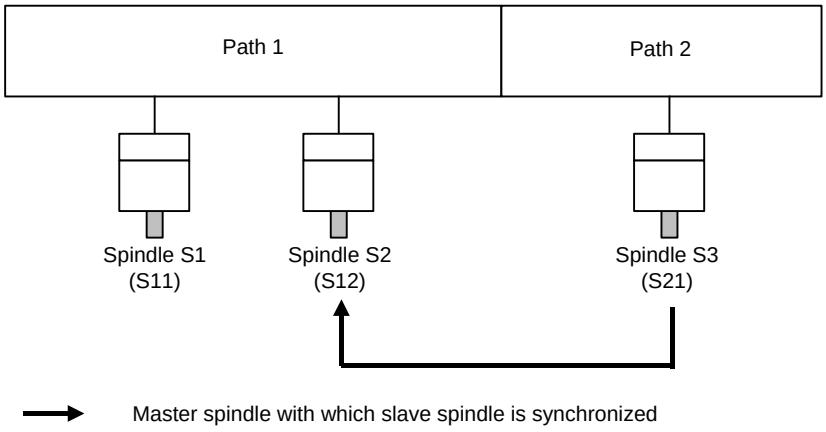
Logic spindle number common to system	Path number + intra-path spindle number
First spindle (S1)	First spindle of path 1 (S11)
Second spindle (S2)	Second spindle of path 1 (S12)
Third spindle (S3)	First spindle of path 2 (S21)



* Example of synchronization of spindles that belong to different paths

Parameter setting
Parameter SSS(No.3704#4) (PATH1)=1, (PATH2)=1
Parameter No. 4831(ALL) =0
Parameter No. 4832(S1) =0, (S2) =0,(S3) =2

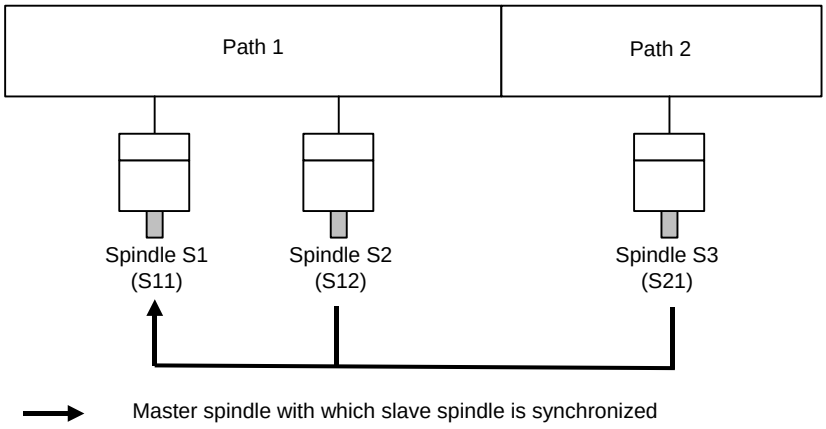
Master spindle	Slave spindle
Second spindle of path 1 (S11)	First spindle of path 2 (S21)



* It is also possible to synchronize more than one slave spindles with the master spindle.

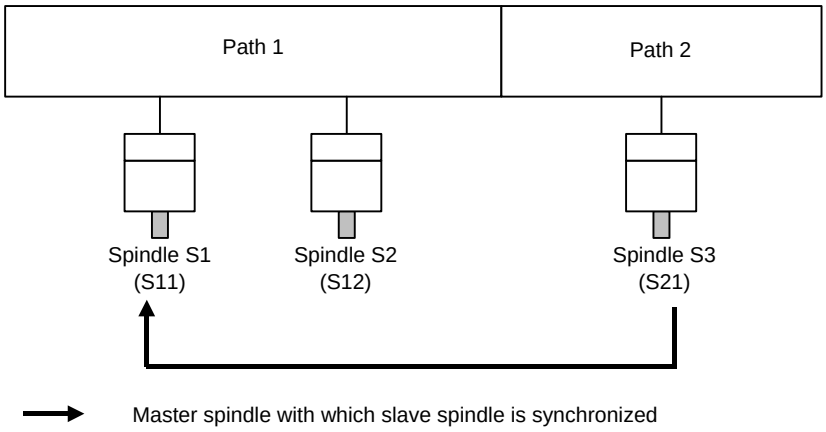
Parameter setting
Parameter SSS (No. 3704#4) (PATH1)=1, (PATH2)=1
Parameter No. 4831(ALL) =0
Parameter No. 4832(S1) =0, (S2) =1,(S3) =1

Master spindle	Slave spindle
First spindle of path 1 (S11)	Second spindle of path 1 (S12)
	First spindle of path 2 (S21)



- When the parameter SCB(No. 4800#5) is set to 1
  The configuration of synchronized spindles is as follows (regardless of the setting of bit 4 (SSS) of parameter No. 3704):

Master spindle	Slave spindle.
First spindle of path 1	First spindle of path 2



As control signals, the 0*i*-TTC system compatible signal interface becomes usable.

- Speed synchronization

- When the spindle synchronous control signal is set to 1, the spindle synchronous control mode is set. When spindle synchronous control is specified, each spindle is accelerated or decelerated to a specified speed then enters the synchronous control state.
- The spindle synchronous speed control completion signal is output when the two spindles have reached the speed matching a specified spindle synchronous speed and the speed difference between the two spindles is within the value set in parameter No. 4033.

- Phase matching

- Spindle phase synchronization is executed by setting the spindle phase synchronization control signal to 1 in the spindle synchronous control mode (after the output of the spindle synchronous speed control completion signal).
  The synchronous state between the two axes is not maintained during spindle phase synchronization operation (until the spindle phase synchronization control completion signal is set to 1).
- The spindle phase synchronization completion signal is output when the error difference between the two spindles lies within the allowable number of pulses set in the NC parameter (No. 4810).

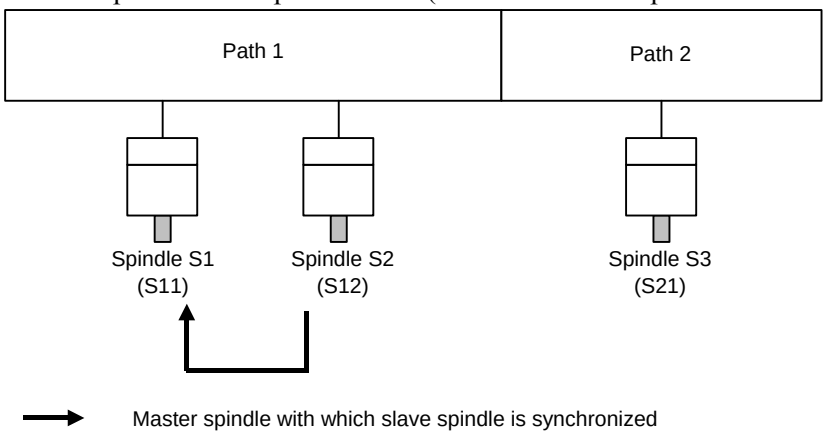
If spindle phase synchronization is specified after spindle phase synchronization completion, another spindle phase synchronization operation is performed.
When the two spindles hold a workpiece, do not specify spindle phase synchronization.

- Speed specification

- In the spindle synchronous control mode, the specification of a speed for the master spindle is valid.
- In the spindle synchronous control mode, a speed for the master spindle can be specified in the same way as for an ordinary S command.
  Moreover, signals such as the PMC-based spindle output control signals (SIND, SSIN, SSGN, and R01I to R12I), *SSTP, and SOR are valid as usual.
  If a speed is specified using a PMC-based spindle output control command, however, the parameters (Nos. 3741, 3742, 3743, and 3744) for specifying the maximum speeds of the individual spindle gears must be correctly set beforehand.
  When 0 is set in the parameter corresponding to a selected gear, the spindle does not make a synchronous rotation.
- An S command specified for a master spindle and a speed specified for PMC-based spindle output control before the spindle synchronous control mode is set are valid even in the spindle synchronous control mode.
  A speed command for a master spindle during spindle synchronous control is also valid as a command for the master spindle after spindle synchronous control is canceled.

- Speed ratio control

- Spindle synchronous control can be exercised to set a speed ratio of master spindle:slave spindle to 1:n.
  Speed ratio control is exercised in spindle synchronization by setting the spindle synchronous speed ratio control signal SBRT <G0038.1> to 1 in the spindle synchronous control mode. At this time, the speed ratio of master spindle:slave spindle is 1:n (n: value set in the parameter SBRATE (No.7635)).



Mater spindle : Speed ratio setting for the slave spindle 1:2
When the speed S100 is specified for master spindle S1
Spindle S1 : Spindle speed S100(1/min)
Spindle S2 : Placed in synchronous control state with spindle speed S200 (1/min).

If the speed of each slave spindle specified by multiplying the speed specified for the master spindle by a speed ratio exceeds the value set in the parameter (No. 7636), the speed of each slave spindle is clamped to a set maximum allowable speed. At the same time, the speed of the master spindle is clamped to the speed that maintains the speed ratio.
When the speeds of multiple slave spindles are clamped, all spindles are clamped to the speeds that can maintain the speed ratios for synchronization control.
When the speed of a slave spindle is clamped, the spindle synchronous speed ratio control clamp signal RSMAX <F0065.2> is set to 1.

NOTE
To use speed ratio control, set bit 2 (SBR) of parameter No. 7603 to 1.

- Supplement

- To monitor an error mutually between two spindles in the spindle synchronous control mode, the phase error monitor signal SYCAL <Fn044.4> (SYCAL1 to SYCAL2 <Fn043.0 to Fn043.1>) is available. A synchronization error between two spindles is monitored at all times. When a synchronization error exceeding the value (absolute number of error pulses) set in parameter No. 4811 is detected, this signal is set to 1. When the synchronization error does not exceed the value set in parameter No. 4811, the signal is set to 0.
- Even when two spindles hold a workpiece in the synchronous control state, constant surface speed control can be exercised. Even if the speed of a spindle changes, however, the synchronous speed changes within the parameter-set range (acceleration/deceleration time constant in spindle synchronous control).
- In spindle synchronous control, the spindle speed offset value (parameter No. 3731) is invalid.

For details of synchronous-spindle connection, see the description of serial spindles.

Limitation

- When spindles are placed in a mode other than a spindle rotation mode such as the Cs contouring control mode, rigid tapping, and simple synchronous control (M series), the spindle synchronous control mode cannot be used. In the spindle rotation mode, switch the mode of the master and slave spindles to the spindle synchronous control mode.
- For spindles placed in the spindle synchronous control mode, commands for Cs contouring control, rigid tapping, simple synchronous control (M series), and so forth cannot be specified.
- To specify an S command with the second spindle used as master spindles, the multi spindle control is required. (When PMC-based spindle output control signal (SINDx) is used, the multi spindle control is not required.)
- The maximum speed in spindle synchronous control depends on the maximum speed (parameter No. 4020) of the spindle motor of the master spindle.
  Example)
  Maximum speed of the spindle motor of a master spindle: 6000 min^{-1}
  Maximum speed of the spindle motor of a slave spindle: 4500 min^{-1}
  In the example above, 6000 min^{-1} is specified as the maximum speed of the master spindle for 12bit = 4095. However, when 6000 min^{-1} is specified in spindle synchronous control, the excessive-speed alarm (spindle unit side alarm AL-07) is issued with the slave spindle.
  So, do not specify a speed exceeding 4500 min^{-1} in the example above.

Diagnosis display

0418	Position deviation of each spindle

When spindles are involved in a position loop, the position deviation of each spindle is indicated.
The unit of a detector used in each mode is used.

0425	Synchronization error of each spindle

The absolute value of the synchronization error of each spindle in spindle synchronization is indicated.
A synchronization error is indicated with a slave spindle.

Spindle screen (spindle synchronous control)

In the spindle synchronous control mode, the spindle screen displays the information below.
"Position deviation" indicates the position deviation of the spindle currently displayed.
"Synchronization error" indicates the absolute value of a synchronization error in spindle synchronization. The value of a synchronization error is displayed on the spindle screen for the slave spindle.
When bit 3 (SVP) of parameter No. 3799 is set to 1, "synchronous error" indicates the peak hold value of a synchronization error.

Actual spindle speed (1/MIN) motor speed (1/MIN)
Actual loop gain (0.01/S)
Spindle position deviation (PLS)
Synchronization error (PLS)

Signal

Spindle synchronous control signal SPSYC<Gn038.2>

[Classification] Input signal

[Function] This signal specifies switching to the spindle synchronous control mode for the first/second spindles.

[Operation] When this signal is set to 1, the spindle synchronous control mode for the first/second spindles is set.
When this signal is set to 0, the spindle synchronous control mode for the first/second spindles is canceled.

This signal is valid only when bit 4 (SSS) of parameter No. 3704 is set to 0.
When bit 5 (SCB) of parameter No. 4800 is set to 1, an address of n = 0 is valid.

Spindle synchronous control signal of each spindle SPSYC1 to SPSYC2 <Gn288.0 to Gn288.1>

[Classification] Input signal

[Function] These signals specify the switching of each spindle to the spindle synchronous control mode.

[Operation] When each of these signals is set to 1, the spindle synchronous control mode with a spindle set as a slave spindle is set.
When each of these signals is set to 0, the spindle synchronous control mode with a spindle set as a slave spindle is canceled.
When SPSYCs is set to 1, spindle synchronization with a spindle set as a slave spindle is performed.

These signals are valid only when bit 4 (SSS) of parameter No. 3704 is set to 1 and bit 5 (SCB) of parameter No. 4800 is set to 0.
Which spindle is synchronized with which spindle is set using parameter No. 4831 or No. 4832.

NOTE
For the correspondence between the signal of each spindle and actual signal address, see the description of "Signal address".

Spindle phase synchronization control signal SPPHS<Gn038.3>

[Classification] Input signal

[Function] This signal specifies the spindle phase synchronization control mode (phase matching) for the first/second spindles.

[Operation] When this signal makes a transition from 0 to 1, spindle phase synchronization control for the first/second spindles is exercised.

This signal is valid when the spindle synchronous control signal SPSYC is set to 1.
Specify this signal after the spindle synchronous speed control completion signal FSPSY is set to 1.
A spindle phase synchronization control operation is started on the rising edge of this signal.
So, phase matching once performed is not lost when this signal is set to 0.
However, when this signal is set to 1 from 0 again, a phase matching operation is performed.

This signal is valid only when bit 4 (SSS) of parameter No. 3704 is set to 0.
Set a phase shift amount in parameter No. 4034.
When bit 5 (SCB) of parameter No. 4800 is set to 1, an address of n = 0 is valid.

NOTE
Before specifying this signal, ensure that the spindle synchronous speed control completion signal FSPSY is set to 1. When the spindle synchronous speed control completion signal FSPSY is set to 0, phase matching operation is not performed.

Spindle phase synchronization control signal of each spindle SPPHS1 to SPPHS2 <Gn289.0 to Gn289.1>

[Classification] Input signal

[Function] These signals specify the spindle phase synchronization control mode (phase matching) for each spindle.

[Operation] When each of these signals is set to 1, spindle phase matching is performed in the spindle synchronous control mode with a spindle set as a slave spindle.
These signals are valid when the spindle synchronous control signal SPSYCs for each spindle is set to 1.
Specify each of these signals after the spindle synchronous speed control completion signal FSPSYs for each spindle is set to 1.
A spindle phase synchronization control operation is started on the rising edge of each of these signals.
So, phase matching once performed is not lost when each of these signals is set to 0.
However, when each of these signals is set to 1 from 0 again, a phase matching operation is performed.

These signals are valid only when bit 4 (SSS) of parameter No. 3704 is set to 1 and bit 5 (SCB) of parameter No. 4800 is set to 0.
Set a phase shift amount in parameter No. 4034.

NOTE
1 Before specifying these signals, ensure that the spindle synchronous speed control completion signal FSPSYx is set to 1. When the spindle synchronous speed control completion signal FSPSYx is set to 0, phase matching operation is not performed.
2 Phase matching in the state where spindles are rotating in synchronism is enabled only for one slave spindle synchronized with one master spindle. When phase matching is to be performed in the state where multiple slave spindles are rotating in synchronism with one master spindle, do not specify spindle phase synchronization control on multiple spindles.

NOTE
3 Before performing phase matching of one master spindle with multiple slave spindles, ensure that all spindles are stopped.
4 For the correspondence between the signal of each spindle and actual signal address, see the description of "Signal address".

Spindle synchronous speed control completion signal FSPSY<Fn044.2>

[Classification] Output signal

[Function] This signal posts that spindle synchronous control (speed synchronization) for the first/second spindles is completed.

[Output cond.] This signal is set to 1 in the following case:

- This signal is output when two spindles have reached the speed equivalent to a specified spindle synchronous speed and the speed difference between the two spindles is equal to or less than the value set in parameter No. 4033 in the spindle synchronous control mode.

This signal is set to 0 in the following cases:

- In the spindle synchronous control mode, two spindles have not reached the speed equivalent to a specified spindle synchronous speed.
- In the spindle synchronous control mode, the speed difference between two spindles is greater than the value set in parameter No. 4033.
- The spindle synchronization control mode is not set.

This signal is valid only when bit 4 (SSS) of parameter No. 3704 is set to 0.
When bit 5 (SCB) of parameter No. 4800 is set to 1, an address of n = 0 is valid.

NOTE
Even if this signal is once set to 1, this signal is set to 0 when the speed difference exceeds the value set in parameter No. 4033 for a cause such as cutting load variation.

Spindle synchronous speed control completion signal for each spindle FSPSY1 to FSPSY2 <Fn288.0 to Fn288.1>

[Classification] Output signal

[Function] These signals post that spindle synchronous control (speed synchronization) with each spindle set as a slave spindle is completed.

[Output cond.] This signal is set to 1 in the following case:

- This signal is output when two spindles have reached the speed equivalent to a specified spindle synchronous speed and the speed difference between the two spindles is equal to or less than the value set in parameter No. 4033 in the spindle synchronous control mode.

This signal is set to 0 in the following cases:

- In the spindle synchronous control mode, two spindles have not reached the speed equivalent to a specified spindle synchronous speed.
- In the spindle synchronous control mode, the speed difference between two spindles is greater than the value set in parameter No. 4033.
- The spindle synchronization control mode is not set.

These signals are valid only when bit 4 (SSS) of parameter No. 3704 is set to 1 and bit 5 (SCB) of parameter No. 4800 is set to 0.

NOTE
1 Even if this signal is once set to 1, this signal is set to 0 when the speed difference exceeds the value set in parameter No. 4033 for a cause such as cutting load variation.
2 For the correspondence between the signal of each spindle and actual signal address, see the description of "Signal address".

Spindle phase synchronization control completion signal FSPPH<Fn044.3>

[Classification] Output signal

[Function] This signal posts that spindle phase synchronization control (phase matching) for the first/second spindles is completed.

[Output cond.] This signal is set to 1 in the following case:

- This signal is output when phase matching is completed with the spindle phase synchronization control signal (when the error pulse difference between the two spindles is equal to or less than the value set in parameter No. 4810) after the two spindles have reached the speed equivalent to a specified spindle synchronous speed in the spindle synchronous control mode.

This signal is set to 0 in the following cases:

- In the spindle synchronous control mode, the phase matching of two spindles is not completed.
- In the spindle synchronous control mode, the error pulse difference between two spindles is greater than the value set in parameter No. 4810.
- The spindle synchronous control mode is not set.
- The spindle phase synchronization control mode is not set.

This signal is valid only when bit 4 (SSS) of parameter No. 3704 is set to 0.
When bit 5 (SCB) of parameter No. 4800 is set to 1, an address of n = 0 is valid.

NOTE
Even if this signal is once set to 1, this signal is set to 0 when the speed difference exceeds the value set in parameter No. 4810 for a cause such as cutting load variation.

Spindle phase synchronization control completion signal of each spindle FSPPH1 to FSPPH2 <Fn289.0 to Fn289.1>

[Classification] Output signal

[Function] These signals post that spindle phase synchronization control (phase matching) with each spindle set as a slave spindle is completed.

[Output cond.] This signal is set to 1 in the following case:

- This signal is output when phase matching is completed with the spindle phase synchronization control signal (when the error pulse difference between the two spindles is equal to or less than the value set in parameter No. 4810) after the two spindles have reached the speed equivalent to a specified spindle synchronous speed in the spindle synchronous control mode.

This signal is set to 0 in the following cases:

- In the spindle synchronous control mode, the phase matching of two spindles is not completed.
- In the spindle synchronous control mode, the error pulse difference between two spindles is greater than the value set in parameter No. 4810.
- The spindle synchronous control mode is not set.
- The spindle phase synchronization control mode is not set.

These signals are valid only when bit 4 (SSS) of parameter No. 3704 is set to 1 and bit 5 (SCB) of parameter No. 4800 is set to 0.

NOTE
1 Even if this signal is once set to 1, this signal is set to 0 when the speed difference exceeds the value set in parameter No. 4810 for a cause such as cutting load variation.
2 For the correspondence between the signal of each spindle and actual signal address, see the description of "Signal address".

Spindle phase error monitor signal SYCAL<Fn044.4>

[Classification] Output signal

[Function] This signal posts that in the spindle synchronous control mode for the first/second spindles, the error pulse difference between the two spindles is greater than a parameter-set value.

[Output cond.] This signal is set to 1 in the following case:

- In the spindle synchronous control mode, the error pulse difference between two spindles is greater than the value set in parameter No. 4811 after spindle synchronous control is completed.

This signal is set to 0 in the following cases:

- The spindle synchronous control mode is not set.
- In the spindle synchronous control mode, the error pulse difference between two spindles is equal to or less than the value set in parameter No. 4811.

This signal is valid only when bit 4 (SSS) of parameter No. 3704 is set to 0.
When bit 5 (SCB) of parameter No. 4800 is set to 1, an address of n = 0 is valid.

Spindle phase error monitor signal for each spindle SYCAL1 to SYCAL2 <Fn043.0 to Fn043.1>

[Classification] Output signal

[Function] These signals post that in the spindle synchronous control mode with each spindle set as a slave spindle, the error pulse difference between two spindles is greater than a parameter-set value.

[Output cond.] This signal is set to 1 in the following case:

- In the spindle synchronous control mode, the error pulse difference between two spindles is greater than the value set in parameter No. 4811 after spindle synchronous control is completed.

This signal is set to 0 in the following cases:

- The spindle synchronous control mode is not set.
- In the spindle synchronous control mode, the error pulse difference between two spindles is equal to or less than the value set in parameter No. 4811.

These signals are valid only when bit 4 (SSS) of parameter No. 3704 is set to 1 and bit 5 (SCB) of parameter No. 4800 is set to 0.

NOTE
For the correspondence between the signal of each spindle and actual signal address, see the description of "Signal address".

Spindle synchronous speed ratio control signal SBRT<Gn038.1>

[Classification] Input signal

[Function] This signal specifies switching to speed ratio control in spindle synchronous control.

[Operation] When this signal is set to 1, speed ratio control is set.

When this signal is set to 0, speed ratio control is canceled.

This signal is common to all slave spindles that belong to the same path.
When bit 5 (SCB) of parameter No. 4800 is set to 1, an address of n = 0 is valid.

Spindle synchronous speed ratio control clamp signal RSMAX<Fn065.2>

[Classification] Output signal

[Function] This signal posts the rotation state of a slave spindle under speed ratio control in spindle synchronous control.

[Output cond.] This signal is set to 1 in the following case:

- The speed of a slave spindle is clamped to the value set in the parameter (No. 7636) under speed ratio control in spindle synchronous control.

This signal is set to 0 in the following cases:

- The speed of any slave spindle is not clamped to the value set in the parameter (No.7636) under speed ratio control in spindle synchronous control.
- The spindle synchronization control mode is not set.

This signal is common to all slave spindles that belong to the same path.
When bit 5 (SCB) of parameter No. 4800 is set to 1, an address of n = 0 is valid.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn038					SPPHS	SPSYC	SBRT	
Gn288							SPSYC2	SPSYC1
Gn289							SPPHS2	SPPHS1
Fn044				SYCAL	FSPPH	FSPSY		
Fn065						RSMAX		
Fn288							FSPSY2	FSPSY1
Fn289							FSPPH2	FSPPH1
Fn043							SYCAL2	SYCAL1

- Signal allocation of spindle-type signals

The signal allocation of spindle-type varies according to the path number to which each spindle belongs and intra-path spindle number.

Here, the address allocation of the spindle synchronous control signal for each spindle SPSYCs, which is a spindle-type signal, is described below.

- When the one path system is used :
  When two spindles of path 1 are used

Logic spindle number	Path number + intra-path spindle number	Signal symbol	Signal address
First spindle (S1)	First spindle of path 1 (S11)	SPSYC1	G0288.0
Second spindle (S2)	Second spindle of path 1 (S12)	SPSYC2	G0288.1

- When the 2-path system is used (T series (2-path control)) :
  When three spindles consisting of two spindles of path 1 and one spindle of path 2 are used

Logic spindle number	Path number + intra-path spindle number	Signal symbol	Signal address
First spindle (S1)	First spindle of path 1 (S11)	SPSYC1	G0288.0
Second spindle (S2)	Second spindle of path 1 (S12)	SPSYC2	G0288.1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3799					SVPs			

[Input type] Parameter input
[Data type] Bit spindle

#3 SVPs As synchronization errors displayed on the spindle screen:
0: Monitor values are displayed.
1: Peak-hold values are displayed,
Spindle synchronization errors are displayed on the side of the spindle that functions as a slave axis in spindle synchronization control.

	#7	#6	#5	#4	#3	#2	#1	#0
3704				SSS				

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#4 SSS Synchronous spindle control by each spindle is:
0: Not performed.
1: Performed.
The master axis and slave axis of synchronous spindle control can be selected from the arbitrary spindles.
The target spindle of synchronous spindle control is specified in parameter No. 4831.
In addition, the following signals affect the control.
- Synchronous spindle signals of each spindle SPSYCs
- Signals of synchronous control of the spindle phase for each spindle SPPHSs

	#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 To use a serial spindle, set bit 5 (SSN) of parameter No. 8133 to 0.
2 A maximum of one analog spindle can be controlled.
3 When using an analog spindle, set it at the end of the spindle configuration.

3717	Spindle amplifier 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 spindle 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.
3: Spindle motor connected to amplifier number 3 is used.

NOTE
When using an analog spindle, set it at the end of the spindle configuration.
(Example)
When there are three spindles in an entire system (two serial spindles and one analog spindle), set the spindle amplifier number (this parameter) of the analog spindle to 3.

	#7	#6	#5	#4	#3	#2	#1	#0
4800			SCB	SYM				

[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.

#4 SYM As the maximum spindle speed in spindle synchronization control:
0: The maximum spindle speed of the master spindle is used.
1: The maximum spindle speed of the master spindle or slave spindle, whichever lower, is used.

#5 SCB The combination of a master spindle and slave spindle for spindle synchronization depends on:

0: Setting of bit 4 (SSS) of parameter No. 3704.
When bit 4 (SSS) of parameter No. 3704 is set to 0
The first spindle and second spindle of each path can be selected as the master spindle and slave spindle, respectively, for spindle synchronization.
When bit 4 (SSS) of parameter No. 3704 is set to 1
A combination of arbitrary spindles of each path can be selected for spindle synchronization.
Set a master spindle for each slave spindle in parameter No. 4831. Set a spindle number of each path.
By setting a spindle number common to the system in parameter No. 4832, an arbitrary spindle that belongs to a different path can be selected as a master spindle for spindle synchronization. Set a spindle number common to the system. Set parameter No. 4831 to 0. Spindle synchronization based on arbitrary spindles must be enabled for the path to which a slave spindle belongs and for the path to which a master spindle belongs.

1: Conventional 0*i*-TTC system compatible specifications.
The first spindle of path 1 and the first spindle of path 2 can be selected as the master spindle and slave spindle, respectively, for spindle synchronization.
As control signals, the signal interface of the 0*i*-TTC system compatible specifications can be used.

	#7	#6	#5	#4	#3	#2	#1	#0
4801								SNDs

[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 SNDs During spindle synchronization control, the rotation direction of each spindle motor is:

0: Same as the specified sign.
1: Opposite to the specified sign.

4810	Error pulse between two spindles when synchronizing phases in the spindle synchronization control mode

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 255

Set an allowable error pulse value between two spindles at phase synchronization time in the spindle synchronization control mode.
This parameter is used to check the completion of phase synchronization performed in the spindle synchronization control mode and to check the phase difference during spindle synchronization control.
When the error pulse value between two spindles become equal to or less than the value set in this parameter, the spindle phase synchronization control completion signals FSPPH<F044.3> and FSPPH1 to 2<Fn289.0 to 1> are set to 1.

4811	Allowable error count for the error pulses between two spindles in the spindle synchronization control mode

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 32767

Set the allowable error count for the error pulses between two spindles in the spindle synchronization control mode.
This parameter is used to check a spindle synchronization error phase difference.
When a spindle synchronization error equal to or greater than the value set in this parameter is detected, the phase error monitor signals SYCAL<F044.4> and SYCAL1 to 2 <Fn043.0 to 1> are set to 1.

4831	Master axis of each slave spindle under spindle synchronous control

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 spindle axes (within a path)

When a spindle is set as a slave spindle in spindle synchronization control on each spindle, set which spindle (master spindle) the slave spindle is to be synchronized with.
Examples of parameter setting)

- When spindle synchronization control is exercised with the first spindle selected as a master spindle and the second spindle selected as a slave spindle
  No.4831(1)=0
  No.4831(2)=1

NOTE
1 This parameter is valid if bit 4 (SSS) of parameter No. 3704 is set to 1.
2 The setting of a slave spindle as a master spindle is invalid.
3 In this parameter, set a spindle number within the same path. When a spindle not belonging to the local path is to be selected as a master spindle for spindle synchronization, set a spindle number common to the system in parameter No. 4832. In such a case, set 0 in this parameter.

4832	Master spindle of each slave spindle under spindle synchronization control (spindle number common to the system)

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 spindle axes (common to the system)

When a spindle is set as a slave spindle in spindle synchronization control on each spindle, set which spindle (master spindle) the slave spindle is to be synchronized with.

NOTE
1 This parameter is valid if bit 4 (SSS) of parameter No. 3704 is set to 1.
Bit 4 (SSS) of parameter No. 3704 must be set to 1 (to enable spindle synchronization based on arbitrary spindles) for the path to which a slave spindle belongs and for the path to which a master spindle belongs.
2 The setting of a slave spindle as a master spindle is invalid.
3 In this parameter, set a spindle number common to the system. When this parameter is used, parameter No. 4831 is set to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
7603						SBR		

[Input type] Parameter input
[Data type] Bit path

#2 SBR For spindle synchronization, speed ratio control is:
0: Not used.
1: Used.

NOTE
1 This parameter sets the speed of the slave spindle to an integral multiple of the speed of the master spindle in the spindle synchronization control function.
2 There is no relation with the polygon machining function.
3 Spindle synchronization control needs to be enabled.
4 Parameters No.7635 and No.7636 need to be set.

7635	Ratio of slave spindle speed in spindle synchronization control

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 9
This parameter sets the ratio of master spindle speed:slave spindle speed (1:n) in spindle synchronization control.

NOTE
This parameter is valid only when bit 2 (SBR) of parameter No. 7603 is set to 1.

7636	Maximum allowable slave spindle speed in spindle synchronization control

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 19999
The speed of the slave spindle under speed ratio control in spindle synchronization control is clamped so that the speed does not exceed the value set in this parameter.

NOTE
1 This parameter is valid only when bit 2 (SBR) of parameter No. 7603 is set to 1.
2 When speed ratio control in spindle synchronization control is used, be sure to set this parameter.
3 When 0 is set, the speed is clamped to 0, disabling rotation under spindle synchronization.

	#7	#6	#5	#4	#3	#2	#1	#0
8133				SYC				

NOTE
When at least one of these parameters is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#4 SYC Spindle synchronization is:
0: Not Used.
1: Used.

Alarm and message

Number	Message	Description
PS0194	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 simple spindle synchronous control mode (M series).

Caution

⚠ CAUTION
1 Output signal SYCAL is used for monitoring a phase shift in synchronous control. The processing performed when a phase shift is detected depends on the specifications determined by the machine tool builder.
2 A gear ratio of 1:1 only is allowed between the spindle and position coder.
3 A CNC system restriction is imposed on the maximum number of spindles.

Reference item

For details of the parameters (No. 4000 to No. 4539), signals, and alarms for the serial spindle control unit, refer to the manuals related to serial spindles listed below.
FANUC AC SPINDLE MOTOR α*i* series PARAMETER MANUAL (B-65280EN)
FANUC SERVO AMPLIFIRER α*i* series DESCRIPTIONS (B-65282EN)
FANUC SERVO MOTOR α*i* series MAINTENANCE MANUAL (B-65285EN)

10.15 SPINDLE ORIENTATION

Overview

This function stops the spindle at a specified position. The spindle can be stopped in either of the following two ways.

- The spindle is stopped by applying a function of the spindle control unit.
- The spindle is stopped using mechanical stop.

NOTE
To use the spindle orientation function, set bit 4 (NOR) of parameter No. 8135 to 0.

Explanation

- Using the spindle control unit

Some spindle control units can position the spindle motor by using sensors and position coders. The CNC itself does not control positioning by using these units.

- Mechanical stop

To mechanically stop the spindle by using, for example, a shot pin, rotate the spindle at a constant low speed and drive the pin into the spindle. The spindle can be rotated at a low constant speed by applying either of the following methods.

- Spindle orientation signal SOR<Gn029.5>
- Spindle output control by the PMC

- Serial spindle orientation by a position coder

Two methods of serial interface spindle orientation are available:

- Spindle orientation by a magnetic sensor
- Spindle orientation by a position coder

Spindle orientation by a position coder is described below.

For the function of spindle orientation by a magnetic sensor, refer to
"FANUC SERVO AMPLIFIRER αi series DESCRIPTIONS (B-65282EN)".

Two methods of spindle orientation by a position coder are available:

- Orientation with the fixed stop position
- Orientation with the stop position set externally

<1> Orientation with the fixed stop position: Bit 0 (ORT) of parameter No. 3729 = 0
By inputting the spindle orientation command signal, the spindle can always be stopped at an orientation stop position set beforehand with a parameter.

<2> Orientation with the stop position set externally: Bit 0 (ORT) of parameter No. 3729 = 1
By externally setting the 12-bit orientation external stop position command signal and inputting the spindle orientation command signal, the spindle can always be stopped at an arbitrary orientation position within one spindle rotation.
The least input increment for a stop position is as follows:

$$\text{Leats input increment} = \frac{360°}{4096} = 0.088$$

Signal

Spindle orientation command signals

ORCMA<Gn070.6> : For first spindle
ORCMB<Gn074.6> : For second spindle

[Classification] Input signal

[Function] These signals specify spindle orientation for serial spindle amplifiers.
For details of the sequence, refer to " FANUC SERVO AMPLIFIRER αi series DESCRIPTIONS (B-65282EN)".

Spindle orientation signals with the stop position externally set

SH00A to SH11A<Gn078,Gn079> : For first spindle
SH00B to SH11B<Gn080,Gn081> : For second spindle

[Classification] Input signal

[Function] This command is used for specifying a stop position with an absolute position within one rotation in the following equation:

$$\text{Stop position (degree)} = \frac{360}{4096} \times \sum_{i=0}^{n} (2^i \times Pi)$$

where Pi = 0 when SHAi = 0, or Pi = 1 when SHAi = 1.
For details of the sequence, refer to " FANUC SERVO AMPLIFIRER αi series DESCRIPTIONS (B-65282EN)".

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn070		ORCMA						
Gn078	SH07A	SH06A	SH05A	SH04A	SH03A	SH02A	SH01A	SH00A
Gn079					SH11A	SH10A	SH09A	SH08A
Gn074		ORCMB						
Gn080	SH07B	SH06B	SH05B	SH04B	SH03B	SH02B	SH01B	SH00B
Gn081					SH11B	SH10B	SH09B	SH08B

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3729								ORTs

[Input type] Parameter input
[Data type] Bit spindle

#0 ORTs When a serial spindle is used, the spindle orientation function of stop position external setting type based on the position coder is:
0: Not performed.
1: Performed.

4031	Stop position in orientation by a position coder (MAIN spindle)
4204	Stop position in orientation by a position coder (SUB spindle)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit

[Valid data range] 0 to 4096
Each of these parameters specifies a stop position in orientation by a position coder.
These parameters are valid when bit 0 (ORT) of parameter No. 3729 is set to 0.

4077	Stop position shift amount in orientation by a position coder

4228	Stop position shift amount in orientation by a position coder (SUB spindle)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] -4096 to 4096
Each of these parameters specifies a stop position shift amount in orientation by a position coder.
This parameter is valid no matter what the setting of bit 0 (ORT) of parameter No. 3729 is.

	#7	#6	#5	#4	#3	#2	#1	#0
8135				NOR				

NOTE
When this parameter is set, the power must be turned off before operation is continued.

[Input type] Parameter input
[Data type] Bit

#4 NOR Spindle orientation is:
0: Used.
1: Not Used.

NOTE
This parameter is valid only when spindle serial output can be used.

Caution

⚠ CAUTION
1 To perform spindle orientation by using the spindle control unit, the signals of the spindle control unit must be used.
To perform serial spindle orientation by using a position coder (to perform serial spindle orientation with the stop position set externally), the serial spindle control unit signals must be used.
2 When the function of spindle orientation with the stop position set externally is used, the parameters for stop positions in orientation by a position coder (No. 4031 and No. 4204) are invalid.

Note

NOTE
Spindle orientation with the spindle positioning function differs from that described in this section. For details, see "Spindle Positioning."

Reference item

Manual name	Item name
FANUC SERVO AMPLIFIRER αi series DESCRIPTIONS (B-65282EN)	Position coder method spindle orientation Magnetic sensor method spindle orientation
FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)	Position coder method spindle orientation Magnetic sensor method spindle orientation

10.16 SPINDLE OUTPUT SWITCHING

Overview

Spindle output switching switches between the two motor windings, one for low speed and the other for high speed, incorporated into the special spindle motors. This ensures that the spindle motor demonstrates stable output characteristics over a wide range.

Since spindle output switching is a function of the spindle control unit, also refer to the manual for the spindle control unit being used.

This section describes the relationship between spindle output switching and the spindle control function in the CNC.

Explanation

- Operation of output-switchable spindle motor

To switch the spindle output characteristics, the windings are usually switched using a relay. Prior to the completion of winding switching, the spindle rotates free from drive power.

Output switching changes the relationship between a speed command, issued from the CNC to the spindle, and the output characteristics of the spindle motor. However, the relationship between the speed command and spindle motor speed is not changed.

- Output switching timing

During actual machining, the spindle is usually controlled in the following way.

<1> Constant spindle speed during cutting, such as milling

<2> Continuously changing spindle speed during cutting, such as in constant surface speed control

<3> Controlling the position loop including the spindle motor during rigid tapping, spindle positioning, Cs contour control, etc.

For applications such as those in (1), we recommend switching the output characteristics for low speed and high speed by using the spindle motor speed detection signal of the spindle control unit.

For applications such as those described in (2) and (3), the spindle must not rotate with no drive power applied during cutting or positioning. It is necessary for the output characteristics to be switched appropriately before machining or for output switching to be masked by using a PMC ladder sequence.

- Output switching and gear switching

Spindle output switching ensures that the spindle motor demonstrates stable characteristics over a wide range, and eliminates the requirement of a mechanical spindle gear switching mechanism.

In creating a PMC ladder sequence for output switching, however, using the gear switching of the CNC's spindle control function may facilitate programming.

Note the following points when using gear switching for CNC spindle control for output switching with a machine tool having no mechanical gear switching mechanism.

- When gear selection output signals, GR2O and GR1O <Fn034.0, 1>, are used (for machining centers in which constant surface speed control is not provided and GTT, bit 4 of parameter No. 3706, is set to 0)
  Set two gears, which are almost the same.
  (Example: Value of parameter No. 3741 = value of parameter No. 3742 - 1, value of No. 3742 = Maximum spindle speed)
  When parameter No. 3741 is equal to parameter No. 3742, the CNC judges that one gear is used, and does not output the GR2O signal.
  If two gears are not set almost similarly, the following parameters related to gear switching points cannot be used.
- The parameters related to gear switching points, SGT, bit 3 of parameter No. 3705, and SGB, bit 2 of parameter No. 3705, parameter Nos. 3761 and 3751 can be used.
- In usual spindle control, depending on the speed at switching points, the speed specified by the spindle speed command may differ slightly from the actual speed in the area where the maximum spindle speed is set to the maximum speed - 1. (This is because the spindle motor speed, specified by the speed command, is calculated based on the settings of parameter Nos. 3741 to 3744.)
  This does not apply to rigid tapping. (Because the machine tool is controlled using the feedback signal from the detector in the position loop.)
- When gear selection input signals, GR1 and GR2 <Gn028.1, 2>, are used (for lathes or machining centers in which constant surface speed control is provided or GTT, bit 4 of parameter No. 3706, is set to 1)
  Parameter settings are read according to the input signal information. Unlike the GR2O and GR1O signals, these signals do not require special parameter settings.
  Example)
  When parameter Nos. 3741 and 3742 are set to the maximum spindle speed.
  Create a PMC sequence that specifies the following.
  For gear 1, set GR1 and GR2 to 0.
  For gear 2, set GR2 to 0 and set GR1 to 1.
  The PMC must determine the switching timing on the basis of some information.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (This manual)	Spindle speed control Rigid tapping
FANUC SERVO AMPLIFIRER αi series DESCRIPTIONS (B-65282EN)	Spindle switching control
FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)	Spindle switching control

10.17 SIMPLE SPINDLE SYNCHRONOUS CONTROL (M SERIES)

M

Overview

Spindles can be synchronized with each other for control.
Two spindles can be combined, one as a master spindle and the other as a slave spindle, so that Cs contouring control, rigid tapping, and normal spindle rotation control can also be used with the slave spindle under control by the master spindle.
This function can be used with serial spindles.
Unlike spindle synchronous control, simple spindle synchronous control does not guarantee spindle operation synchronization between a master spindle and slave spindle. Moreover, applicable spindle functions differ.
The major differences from spindle synchronous control are indicated below.

For details, see also Section 11.14, "SPINDLE SYNCHRONOUS CONTROL".

Function		Simple spindle synchronous control	Spindle synchronous control
Combination with other spindle functions	Spindle rotation control	Δ(Same speed command is used.)	O(Synchronization/phase matching is possible.)
	Cs contour control	O	X
	Rigid tapping	O	X
Parking function		O	X

NOTE
Simple spindle synchronous control function is optional function.

Explanation

When the simple spindle synchronous control signal is input from the PMC, the simple spindle synchronous control mode is selected.
In simple spindle synchronous control, a command for the master spindle also functions as a command for the slave spindle.
The ordinary specification method for a master spindle can be used without modification in the spindle rotation control mode, Cs contouring control mode, rigid tapping mode, and spindle positioning mode.
The parking function can stop each spindle under simple spindle synchronous control.

- Configuration of synchronized spindles

In simple spindle synchronous control, a spindle that accepts an S command/move command is referred to as a master spindle, and a spindle that ignores a command for itself and performs operation in synchronism with the master spindle is referred to as a slave spindle.

When bit 5 (SSY) of parameter No. 3704 is set to 0
The configuration of spindles placed under simple spindle synchronous control is as follows:

	Master spindle	Slave spindle
Spindle number	First spindle	Second spindle

When bit 5 (SSY) of parameter No. 3704 is set to 1
Which slave spindle is placed under simple spindle synchronous control with which master spindle is set using parameter No. 4821.

	Master spindle	Slave spindle
Spindle number	Second spindle	First spindle

- Operation in each control mode

1 Spindle rotation control mode
The slave spindle rotates upon the issue of the same command as that used for the master spindle.
The command does not specify the speed of the spindle, instead specifies the ratio of the spindle motor speed to the maximum speed. So, only when the spindle unit configuration of master and slave spindles (parameter settings for a maximum motor speed, motor-spindle gear ratio, and so forth) is the same, the two spindles rotate at about the same speed. (However, the synchronous state is not guaranteed.)
For details of spindle rotation control, see “SPINDLE SPEED CONTROL.”

2 Cs contour control mode
When the Cs contouring control is used together, Cs contouring control on a slave spindle operating in synchronism with the master spindle is possible. (Control independent of the master spindle is impossible.)

A move command address and position are displayed only for the master spindle.
In reference position return operation, the two spindles are checked for reference position return completion before the reference position return completion signal is turned 1.
In simple spindle synchronous control, switching between the spindle rotation control mode and the Cs contouring control mode is possible.
For details of Cs contouring control, see "Cs CONTOUR CONTROL".

3 Rigid tapping
When the rigid tapping option is selected, the slave spindle supports rigid tapping under the control of a rigid tapping command for the master spindle. (Rigid tapping for the slave spindle cannot be performed independently of the master spindle.)
For details of rigid tapping, see "RIGID TAPPING."

- Parking function

The parking function stops the rotation and movement of each of master and slave spindles placed in the simple spindle synchronous control mode, regardless of the spindle control mode (spindle rotation control, Cs contouring control, or rigid tapping).
In the parking state, the spindle is placed in the following state, depending on its mode:

Spindle rotation control mode:
Rotation is stopped in the same way as when S0 is specified.
Other modes:
Movement is stopped in the same way as in the case of no distribution pulse.

With a move command for positioning of a Cs contouring control axis or spindle in the parking state, only the movement of the spindle in the parking state is stopped by updating the coordinate system.

Reference position return operation for a Cs contouring control axis and spindle orientation operation for spindle positioning are disabled for a spindle placed in the parking state.
If reference position return operation in the Cs contouring control mode or spindle orientation in the spindle positioning mode is specified with one spindle placed in the parking state, the reference position return completion signal is turned 1 when the spindle not placed in the parking state has completed a reference position return operation.

The parking function stops the movement of only a spindle that is placed in the parking state. So, the parking function can be used, for example, to continue machining by stopping one spindle no longer required or to perform Cs contouring control, rigid tapping, and spindle positioning with the slave spindle by placing the master spindle in the parking state.

NOTE
1 In the simple spindle synchronous control mode, the two spindles are checked for a position error at all times, regardless of the parking state. The excessive error alarm is issued when a spindle has moved due to a load from the position where the spindle was stopped by the parking function.
2 The parking function stops the movement of only a spindle that is placed in the parking state. So, if parking is applied to a spindle after a reference position return operation or spindle orientation operation is once performed, the spindle position is shifted from the machine coordinates. Until another reference position return operation or spindle orientation operation is performed after parking is canceled, the relationship between the spindle position and machine coordinates is not guaranteed.

NOTE
3 The parking signal becomes immediately valid in the simple spindle synchronous control mode. However, even if the parking signal is input during reference position return operation in the Cs contouring control mode or during spindle orientation operation, the reference position return operation or spindle orientation operation continues until it is completed. Parking starts immediately after completion of such an operation.
4 Do not specify a reference position return operation in the Cs contouring control mode or a spindle orientation operation while the two spindles are placed in the parking state. When the two spindles are placed in the parking state, the reference position return operation cannot be completed.

- Simple spindle synchronization and phase error monitor signal

Simple spindle synchronous control does not guarantee synchronous spindle operation. In spindle control involving a position loop as in the case of Cs contouring control and rigid tapping, however, the synchronous state of the two spindles can be maintained by matching the position gains of the two spindles.

For monitoring of the synchronous state of two spindles in such a case, the phase error monitor signal SYCAL is available. The error between two spindles is monitored at all times. When a synchronization error exceeding the value set in parameter No. 4826 is detected, this signal is set to 1. When a synchronization error not exceeding the value set in parameter No. 4826 is detected, this signal is set to 0.

In the spindle rotation control mode, however, synchronous state monitoring is not performed. So, the phase error monitor signal has no effect, and is set to 0 at all times.

Note that even if phase error monitor signal SYCAL is output, no CNC alarm is issued. This signal is designed to be used by the PMC to monitor the machine synchronization state. Its usage varies depending on the machine tool builder. The use of this signal is not mandatory.

NOTE
During simple spindle synchronous control, a reference position return operation in the Cs contouring control mode, a spindle orientation operation in the spindle positioning mode, or a function (such as spindle orientation) that is directly specified for a spindle with a PMC signal is performed for each spindle independently, so that synchronization is not guaranteed.

- Relationship between simple spindle synchronous control and spindle synchronous control

The spindle synchronous control mode and the simple spindle synchronous control mode cannot be selected for the same spindle at the same time.
In a mode other than spindle synchronous control mode, no restriction is imposed on the selection timing of simple spindle synchronous control. Usually, select simple spindle synchronous control when the master spindle is not executing a move command in the spindle rotation control mode or in the Cs contouring control mode.
No restriction is imposed on the timing of simple spindle synchronous control cancellation. Usually, cancel simple spindle synchronous control when no command is executed in any spindle control mode.
When simple spindle synchronous control is canceled, the mode of the slave spindle switches to the spindle rotation control mode. The master spindle maintains the selected spindle control mode.

The table below indicates the spindle control mode switching of the master and slave spindles based on the transitions of the simple spindle synchronous control signal and spindle synchronous control signal.

Spindle control mode switching	Switching when the simple spindle synchronous control signal makes a transition from 0 to 1 in each spindle control mode selection state			
Master spindle	SP → SP(ESY)	CT → CT(ESY)	SV → SV(ESY) Note 3	SY → SY Note 4
Slave spindle	SP → SP(ESY) Note 1	SP → CT(ESY) Note 2	SP → SV(ESY) Note 3	SY → SY Note 4

Spindle control mode switching	Switching when the simple spindle synchronous control signal makes a transition from 0 to 1 in each simple spindle control mode selection state		
Master spindle	SP(ESY) → SP	CT(ESY) → CT	SV(ESY) → SV
Slave spindle	SP(ESY) → SP	CT(ESY) → SP	SV(ESY) → SP

Spindle control mode switching	Switching when the spindle synchronous control signal makes a transition from 0 to 1 in each simple spindle control mode selection state		
Master spindle	SP(ESY) → SP(ESY) Note 4	CT(ESY) → CT(ESY) Note 4	SV(ESY) → SV(ESY) Note 4
Slave spindle	SP(ESY) → SP(ESY) Note 4	CT(ESY) → CT(ESY) Note 4	SV(ESY) → SV(ESY) Note 4

The symbols used in the tables above have the following meanings:

SP : Spindle rotation control mode
CT : Cs contour control mode
SV : Rigid tapping mode or spindle positioning mode
SP(ESY) : Spindle rotation control mode during simple spindle synchronous control
CT(ESY) : Cs contouring control mode during simple spindle synchronous control
SV(ESY) : Rigid tapping or spindle positioning mode during simple spindle synchronous control
SY : Spindle synchronous control mode

NOTE

1 After initialization to the spindle rotation control mode, operation is performed with a speed command for the master spindle. So, a speed change can occur.
2 Even if the master spindle is already at the reference position, the slave spindle is not at the reference position yet when the mode of the slave spindle is just switched to the Cs contouring control mode by simple spindle synchronous control. So, perform a reference position return operation again.
3 When the master spindle is placed in the rigid tapping, the input of the simple spindle synchronous control signal is masked in the CNC, so that the mode is not switched to the simple spindle synchronous control mode. After the rigid tapping or spindle positioning mode of the master spindle is canceled, the master spindle and slave spindle are placed in the simple spindle synchronous control mode. After the simple spindle synchronous control mode is once set, the specification or cancellation of the rigid tapping of the master spindle is applied to both of the master spindle and slave spindle.
4 The spindle control mode currently selected is maintained. As an alarm indicating that the spindle control mode is not switched, alarm (PS0194) is displayed. This alarm can be cleared by resetting the signal input later to 0.
5 Switch to the simple spindle synchronous control mode when the slave spindle is placed in the spindle rotation control mode.
6 For the slave spindle placed in the simple spindle synchronous control mode, the switching to a spindle mode such as Cs contouring control, rigid tapping, and spindle synchronous control cannot be specified.

- Spindle rotation direction

In the simple spindle synchronous control mode, the same speed command and move command are issued to the master spindle and slave spindle.
The specified rotation direction of a spindle can be changed using a PMC signal (spindle forward/reverse rotation command signal SFR or SRV) or serial spindle parameter according to the control mode.

- Position deviation check and alarm display

When simple spindle synchronous control is used in a mode other than the spindle rotation control mode, the master spindle and slave spindle are checked for a position deviation.
An in-position check is made to see if the position deviation of each of the two spindles is within a parameter-set range.
Position deviation excessive error checks at stop time and move time are made to see if the position deviation of each spindle exceeds a parameter-set value.
The parameters set for the master spindle are used for both spindles.
An alarm for an excessive error is displayed for the master spindle. Which of the master spindle and slave spindle has entered the excessive error state cannot be identified on the screen.

Limitation

- Both of the master spindle and slave spindle must have hardware such as a spindle-related detector required for the functions (Cs contouring control and rigid tapping) that use simple spindle synchronous control.
- When S command is specified using the second spindle as master spindle, the multi-spindle control option is required. (When PMC-based spindle output control is used, the multi-spindle control option is not required.)

Diagnosis display

418	Position deviation of each spindle

When spindles are involved in a position loop, the position deviation of each spindle is indicated.
The unit is pulses. The detection unit per pulse varies from one spindle control mode to another.

425	Synchronization error of each spindle

The absolute value of the synchronization error of each spindle in simple spindle synchronous control is indicated.
A synchronization error is indicated with a slave spindle.
In the spindle rotation control mode, the synchronization error is 0 at all times.

Spindle screen

In the simple spindle synchronous control mode, the display dedicated to the simple spindle synchronous control mode is not provided. Instead, a spindle control mode selected in the simple spindle synchronous control mode is displayed.
When the Cs contouring control is set in simple spindle synchronous control, the Cs contouring control mode is displayed.

Signal

Simple spindle synchronous control signal ESRSYC<Gn064.6>

[Classification] Input signal

[Function] This signal specifies switching to the simple spindle synchronous control mode for the first/second spindles.

[Operation] When this signal is set to 1, the simple spindle synchronous control mode for the first/second spindles is set.
When this signal is set to 0, the simple spindle synchronous control mode for the first/second spindles is canceled.
This signal is valid only when bit 5 (SSY) of parameter No. 3704 is set to 0.

Simple spindle synchronous control signal for each spindle
ESSYC1<Gn264.0> : For first spindle
ESSYC2<Gn264.1> : For second spindle

[Classification] Input signal

[Function] These signals specify switching to the simple spindle synchronous control mode for each spindle.

[Operation] When each of these signals is set to 1, the simple spindle synchronous control mode with the corresponding spindle set as a slave spindle is set.
When each of these signals is set to 0, the simple spindle synchronous control mode with the corresponding spindle set as a slave spindle is canceled.
When ESSYC1 is set to 1:
Simple spindle synchronous control with the first spindle set as a slave spindle is exercised.
When ESSYC2 is set to 1:
Simple spindle synchronous control with the second spindle set as a slave spindle is exercised.
These signals are valid only when bit 5 (SSY) of parameter No. 3704 is set to 1.
Which spindle is synchronized with which spindle is set using parameter No. 4821.

First simple spindle synchronous parking signal PKESS1<Gn122.6> or PKESS1<Gn031.6>

[Classification] Input signal

[Function] This signal parks the first spindle in simple spindle synchronous control.

[Operation] When this signal is set to 1, the first spindle is placed in the parking state.
When this signal is set to 0, the parking state of the first spindle is canceled.
When bit 7 (SPK) of parameter No. 4800 is set to 1, <Gn031.6> is used.
This signal is valid only when bit 5 (SSY) of parameter No. 3704 is set to 0.

Second simple spindle synchronous parking signal PKESS2<Gn122.7> or PKESS2<Gn031.7>

[Classification] Input signal

[Function] This signal parks the second spindle in simple spindle synchronous control.

[Operation] When this signal is set to 1, the second spindle is placed in the parking state.
When this signal is set to 0, the parking state of the second spindle is canceled.
When bit 7 (SPK) of parameter No. 4800 is set to 1, <Gn031.7> is used.
This signal is valid only when bit 5 (SSY) of parameter No. 3704 is set to 0.

Simple spindle synchronous parking signal for each spindle
PKESE1<Gn265.0> : For first spindle
PKESE2<Gn265.1> : For second spindle

[Classification] Input signal

[Function] These signals park each spindle in simple spindle synchronous control.

[Operation] When each of these signals is set to 1, the corresponding spindle is placed in the parking state.
When each of these signals is set to 0, the parking state of the corresponding spindle is canceled.
When PKESE1 is set to 1: The first spindle is placed in the parking state.
When PKESE2 is set to 1: The second spindle is placed in the parking state.
This signal is valid only when bit 5 (SSY) of parameter No. 3704 is set to 0.

Phase error monitor signal SYCAL<Fn044.4>

[Classification] Output signal

[Function] This signal posts that in the simple spindle synchronous control mode for the first/second spindles, the error pulse difference between the two spindles is greater than a parameter-set value.

[Output cond.] This signal is set to 1 in the following case:

- In the simple spindle synchronous control mode, the error pulse difference between two spindles is greater than the value set in parameter No. 4826 after simple spindle synchronous control is completed.

This signal is set to 0 in the following cases:

- The simple spindle synchronous control mode is not set.
- In the simple spindle synchronous control mode, the error pulse difference between two spindles is equal to or less than the value set in parameter No. 4826.

This signal is valid only when bit 5 (SSY) of parameter No. 3704 is set to 0.

Phase error monitor signal for each spindle
SYCAL1<Fn043.0> : For first spindle
SYCAL2<Fn043.1> : For second spindle

[Classification] Output signal

[Function] These signals post that in the simple spindle synchronous control mode with each spindle set as a slave spindle, the error pulse difference between two spindles is greater than a parameter-set value.

[Output cond.] Each of these signals is set to 1 in the following case:

- In the simple spindle synchronous control mode, the error pulse difference between two spindles is greater than the value set in parameter No. 4826 after simple spindle synchronous control is completed.

Each of these signals is set to 0 in the following cases:

- The simple spindle synchronous control mode is not set.
- In the simple spindle synchronous control mode, the error pulse difference between two spindles is equal to or less than the value set in parameter No. 4826.

These signals are valid only when bit 5 (SSY) of parameter No. 3704 is set to 1.

Even in the simple spindle synchronous control mode, the signals that need to be input/output or are used to control those functions (such as spindle orientation) directly specified from the PMC for the spindle control unit are provided independently for each spindle.
For first spindle : (DI:G0070 to G0073, DO:F0045 to F0048)
For second spindle :(DI:G0074 to G0077, DO:F0049 to F0052)
When using the simple spindle synchronous control function, exercise control on the simple spindle synchronous control signals and also manipulate the signals for each spindle as required.
(Such a signal state that the master spindle and slave spindle under simple spindle synchronous control can make the same movement needs to be set.)

NOTE
In the simple spindle synchronous control mode, the states of both the master spindle and slave spindle are checked. So, if simple spindle synchronous control is specified in the Cs contouring control mode before the slave spindle is not activated yet, for example, the VREADY OFF alarm may be issued. Pay attention to control on the PMC signals for the slave spindle as well.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn031	PKESS2	PKESS1						
Gn064		ESRSYC						
Gn122	PKESS2	PKESS1						
Gn264							ESSYC2	ESSYC1
Gn265							PKESE2	PKESE1
Fn043							SYCAL2	SYCAL1
Fn044				SYCAL				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3704								
			SSY					

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#5 SSY Simple synchronous spindle control by each spindle is:
0: Not performed.
1: Performed.
The master axis and slave axis of simple synchronous spindle control can be selected from the arbitrary spindles.
The target spindle of simple synchronous spindle control is set in parameter No. 4821.
In addition, the following signals affect the control.
- Signals of simple synchronous control of each spindle ESSYCs
- Parking signals of simple synchronous control of each spindle PKESEs

	#7	#6	#5	#4	#3	#2	#1	#0
4800	SPK	EPZ						

[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.

#6 EPZ When the parking signal is switched in the reference position established state during Cs contour control exercised using simple spindle synchronous control:
0: Reference position established state is continued.
1: Reference position established state is canceled.
If this parameter is set, the same reference position return operation as manual reference position return is performed with the G28 command immediately after the parking signal is switched.
The G00 command performs a positioning operation including reference position return (when bit 1 (NRF) of parameter No. 3700 is set to 0).

#7 SPK As the parking signals for simple spindle synchronous control:
0: PKESS1<Gn122.6> (first spindle) and PKESS2<Gn122.7> (second spindle) are used.
1: PKESS1<Gn031.6> (first spindle) and PKESS2<Gn031.7> (second spindle) are used.

NOTE
This parameter is valid only when bit 5 (SSY) of parameter No. 3704 is set to 0.

	#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 To use a serial spindle, set bit 5 (SSN) of parameter No. 8133 to 0.
2 A maximum of one analog spindle can be controlled.
3 When using an analog spindle, set it at the end of the spindle configuration.

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.
3: Spindle motor connected to amplifier number 3 is used.

NOTE
When using an analog spindle, set it at the end of the spindle configuration.
(Example)
When there are three spindles in an entire system (two serial spindles and one analog spindle), set the spindle amplifier number (this parameter) of the analog spindle to 3.

4821 **Master axis of each slave spindle under simple synchronous spindle control**

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 (within a path)
When a spindle is set as a slave spindle in simple spindle synchronous control on each spindle, set which spindle (master spindle) the slave spindle is to be synchronized with.

Examples of parameter setting)
- When simple spindle synchronous control is exercised with the first spindle selected as a master spindle and the second spindle selected as a slave spindle
  No.4821(1)=0
  No.4821(2)=1

NOTE
This parameter is valid if bit 5 (SSY) of parameter No. 3704 is set to 1.

4826 **Allowable error count for the error pulses between two spindles in the simple synchronization spindle control mode**

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] 0 to 32767
Set the allowable error count for the error pulses between two spindles in the simple synchronization spindle control mode.
This parameter is used to check a spindle synchronization error phase difference.
When a spindle synchronization error equal to or greater than the value set in this parameter is detected, the spindle phase error monitor signals SYCAL<Fn044.4> and SYCALs are set to 1.

NOTE
1 The detection unit per pulse depends on the spindle control mode (Cs contour control, rigid tapping, or spindle positioning).
2 Set this parameter for a spindle that is to function as a slave spindle. Set 0 for the master spindle.
3 In the spindle rotation control mode, synchronization error detection is not performed.

Alarm and message

Number	Message	Description
PS0194	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 simple spindle synchronous control mode.

Caution

⚠ **CAUTION**
As an output signal for monitoring a phase error during simple spindle synchronous control, the phase error monitor signal SYCAL is available. The action to be taken when a phase error is detected depends on the specifications of the machine tool builder.

Reference item

For details of the parameters (No. 4000 to No. 4539), signals, and alarms for the serial spindle control unit, refer to the manuals related to serial spindles listed below.
FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)
FANUC SERVO AMPLIFIRER αi series DESCRIPTIONS (B-65282EN)
FANUC SERVO SYSTEM αi series MAINTENANCE MANUAL (B-65285EN)

10.18 SPINDLE SPEED FLUCTUATION DETECTION (T SERIES)

T

Overview

With this function, an overheat alarm (OH0704) 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.

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))

Parameter FLR (No.4900#0)= 0	Parameter FLR (No.4900#0)= 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:

<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 (q = parameter (No.4911))

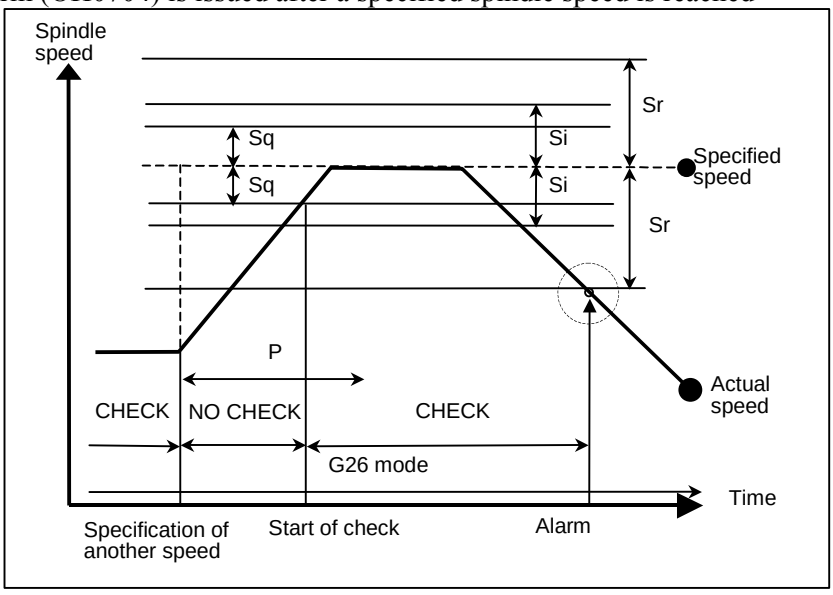
Parameter FLR= 0	Parameter FLR= 1
$Sq = Sc \times \frac{q}{100}$	$Sq = Sc \times \frac{q}{1000}$

<2> When time p specified in parameter No. 4914 elapses after the specified speed Sc changes.

- Examples of spindle speed fluctuation detection

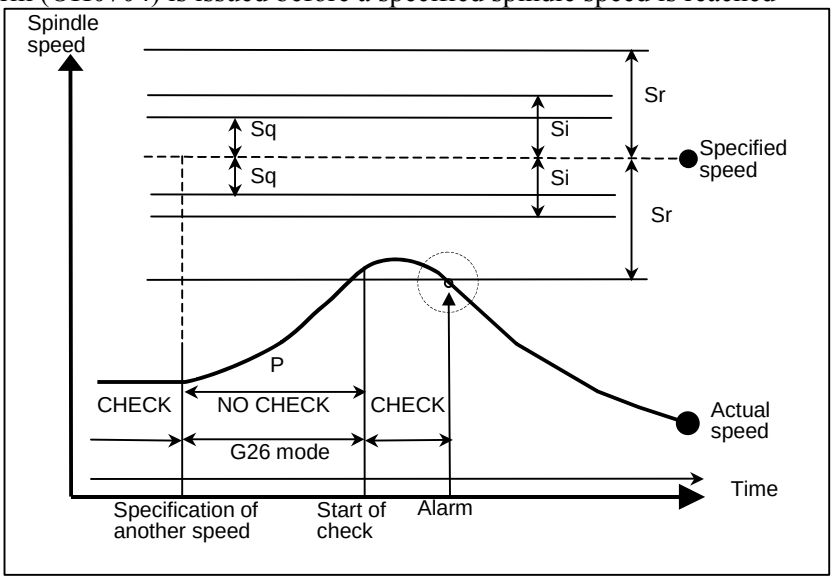
(Example 1)

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

Spindle / Function	Serial spindle	
	1st spindle	2nd spindle
Spindle speed fluctuation detection	Possible	Possible(*1)

Spindle / Function	Analog spindle	
	1st spindle	
Spindle speed fluctuation detection	Possible	

NOTE
1 Multi spindle control function 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<Gn028.7>)
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 that can be subjected to the spindle speed fluctuation detection function, depending on the relationship between system configuration and spindle

The table below show some examples explaining the spindle to be subjected to the spindle speed fluctuation detection function, depending on the system configuration and spindle assignment.

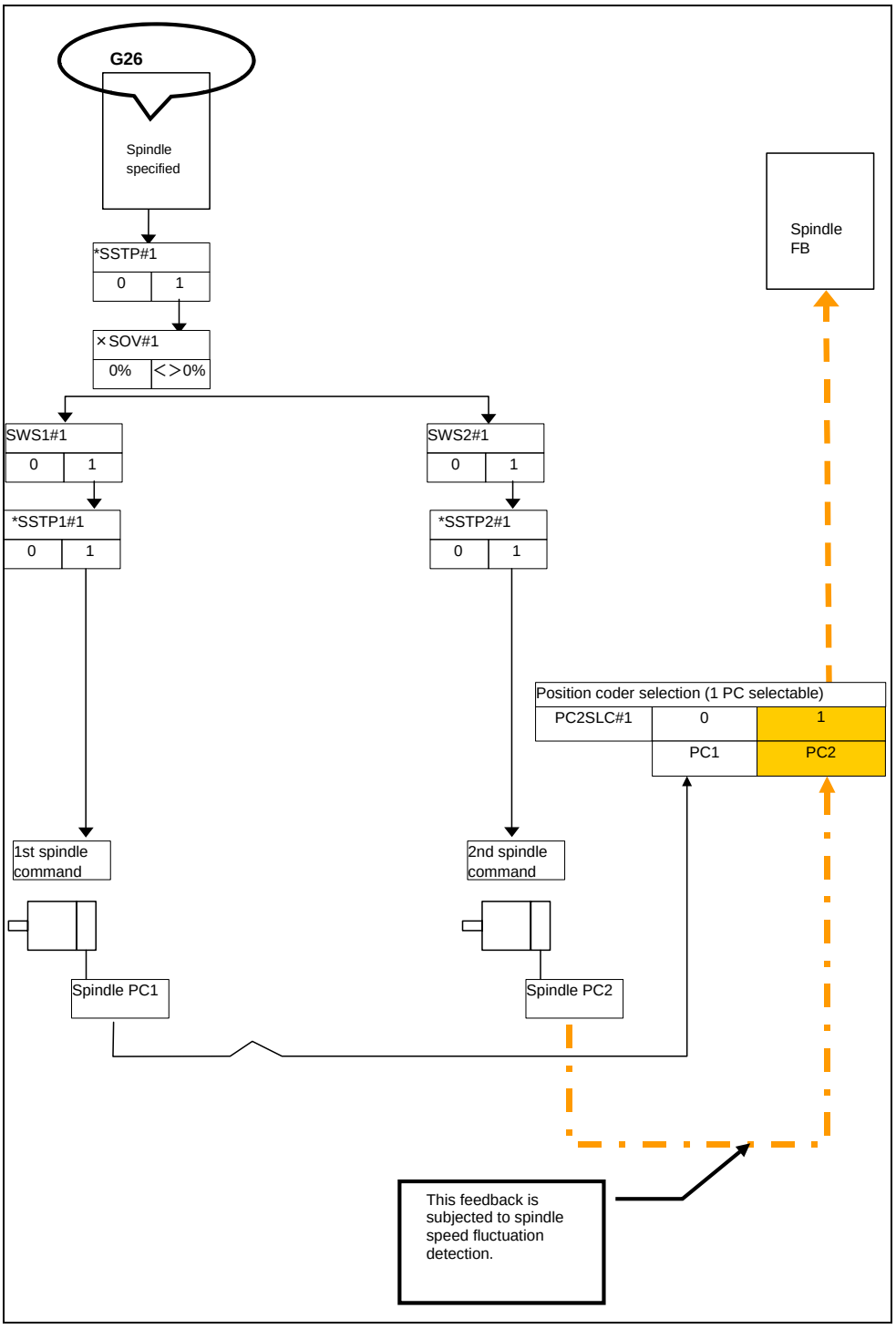
NOTE
1 If path control is performed, the spindle speed fluctuation detection function is carried out for the axis on which the currently selected position coder is mounted.
* Position coder selection signals (PC2SLC<Gn028.7>)
2 Under 2-path control, if the feedback pulse of a position coder mounted on a spindle belonging to a path is used, the selected path feedback must be considered in addition to the selection of the position coder. Great attention should be given to the selection statuses of relative signals.
* Path spindle feedback selection signals (SLPCA<Gn064.2>, SLPCB<Gn064.3>)
3 It is assumed that the reader has understood the individual functions described in this manual. If necessary, see the sections of "spindle control for each path," "spindle control," and "multi spindle."

Examples of spindles subjected to detection

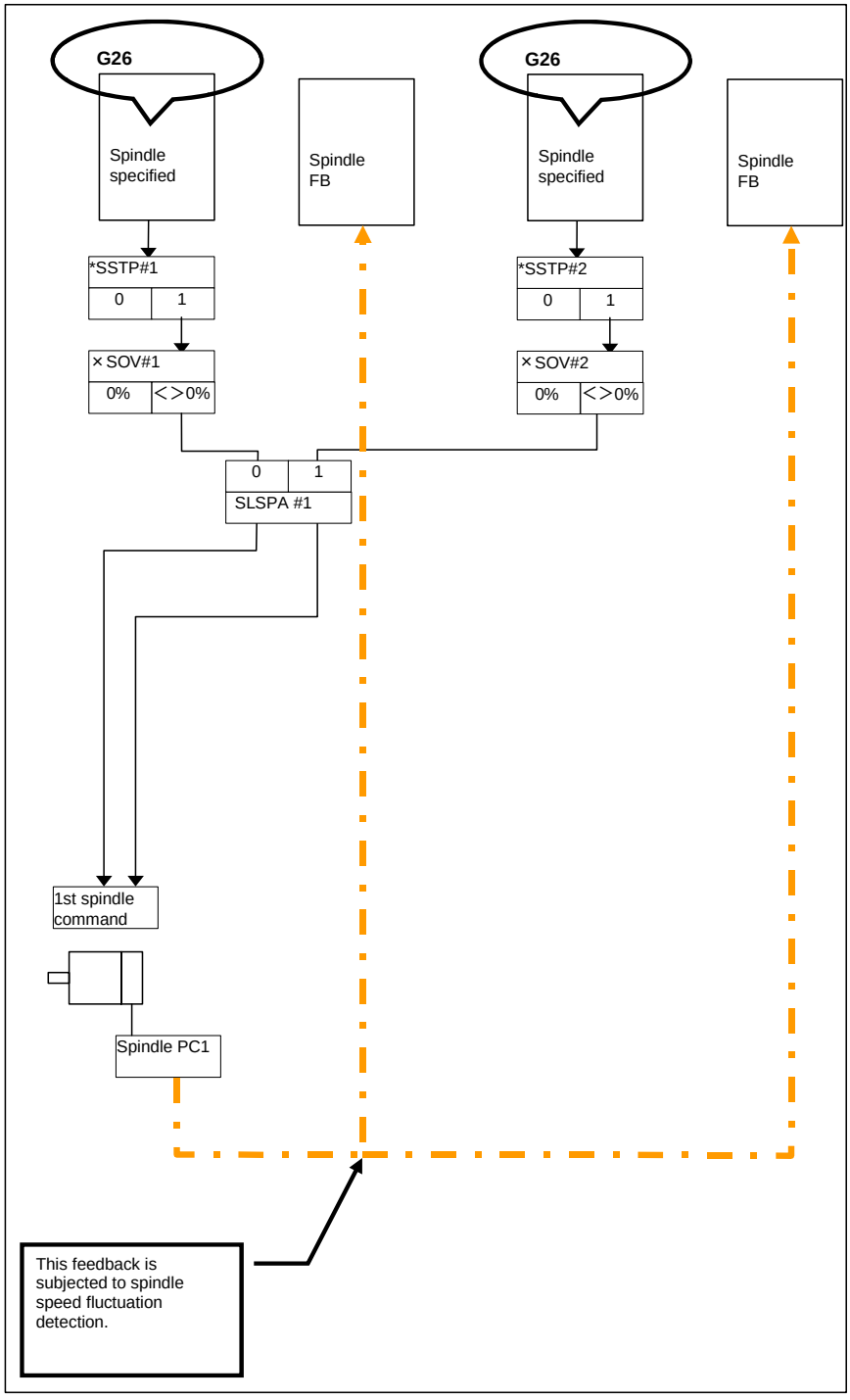
- Specified spindle : Indicates which path is specifying which spindle.
- Feedback pulse : Indicates the contents of the feedback pulse to be used, depending on a selected position coder.
- Feedback subjected to detection : Indicates the feedback subjected to the spindle speed fluctuation detection function.
- G26 : Indicates whether the spindle speed fluctuation detection function is enabled in which path.
- Detection : Indicates whether spindle speed fluctuation detection is possible in which path.

System configuration	Path	Specified spindle	Feedback pulse	Feedback subjected to detection	G26 (Multi-path)	Detection
Single path control + Non-multi spindle control + 1 spindle	——	Address S—> 1st spindle	Feedback<—— 1st spindle PC1	<—O	O	O
Single path control + Multi spindle control + 2 spindles	——	Address S—> 1st spindle —> 2nd spindle	Feedback<— 1st spindle PC1 —— 2nd spindle PC2	<—× <—O	O	O
2-path control + Non-multi spindle control + 1 spindle per path	Signal of spindle control for each path: Type A (*1.)					
	Path 1	Address S—> 1st spindle	Feedback<— 1st spindle PC1	<—O	O	O
	Path 2		Feedback<—	—	O	O
	Signal of spindle control for each path: Type A (*1.)					
	Path 1	Address S—> 1st spindle	Feedback<— 1st spindle PC1	<—O	×	×
	Path 2		Feedback<—	—	O	O

*1 The feedback pulse can be obtained from path 1 in this example.



System controlling two spindles by one path



System controlling one spindle by two paths

Signal

Spindle fluctuation detection alarm signal SPAL <Fn035.0>

[Classification] Output signal

[Function] This signal indicates that the actual spindle speed is not within a tolerance to the specified speed.

[Output cond.] The signal becomes logical "1" when:

- The actual spindle speed goes out of tolerance to the specified speed.

The signal becomes logical "0" when:

- No alarm (OH0704) condition has been detected for spindle speed fluctuation.
- An alarm condition is cleared by resetting the NC when the signal is logical "1".

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn035								SPAL

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
4900								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 parameter No. 4911 and No. 4912 is:
0: 1%
1: 0.1%

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 variation 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 99999

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.

Alarm and message

Number	Message	Description
OH0704	OVERHEAT:SPINDLE	Spindle is overheated by the spindle speed fluctuation detection function. <1>For heavy cutting, ease the cutting conditions. <2>Check whether the cutter has been blunted. <3>Check whether the spindle amplifier is malfunctioning.
PS5305	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 parameter EMS (No. 3702#1) 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.

Note

NOTE

1 Even if the conditions to start the spindle speed fluctuation detection function are satisfied, the spindle speed fluctuation detection function is not performed under the following conditions:
- The specified spindle speed is 0 min^{-1}.
- A search by program restart is in progress.

2 If the spindle stop signal *SSTP<Gn029.6> is set to 0, detection is not performed.

3 If any of the spindle stop signals *SSTP1 to *SSTP2<Gn027.3,4> is set to 0, detection is not performed on the corresponding spindle.

NOTE

4 If alarm OH0704 is raised, a single-block stop occurs in automatic operation. The spindle overheat alarm is displayed on the screen. In addition, the spindle speed fluctuation detection alarm signal (SPAL) is output (the signal is set to 1 in the alarm state).
This signal is cleared to 0 by a reset.

5 If the cause of alarm OH0704 is not eliminated by a reset, the alarm will be raised again.

6 Alarm OH0704 can be suppressed across an allowable variation width specified in parameter No.4913. If the actual spindle speed continues to be 0 min^{-1} for one second or longer, however, an alarm will be raised.

7 The raising of alarm OH0704 does not automatically cause the spindle to stop.

8 G25 and G26 must be specified in a predetermined format. Avoid specifying the preparatory function, auxiliary functions, and the like together with G25 or G26.

9 A serial spindle signal that can affect the actual spindle speed should be manipulated in the G25 mode.

10 If a spindle is under velocity control, spindle speed fluctuation detection is carried out.

11 Avoid making a spindle orientation operation while the spindle speed fluctuation detection function is enabled.

12 If address P is used to select a spindle while the multi spindle function is carried out, avoid specifying S_P_; together with G26. If this is attempted, alarm PS5305 will be raised.

13 For a serial spindle, carefully check the gear count selection statuses of clutch/gear signals (CTH1, CTH2) and gear selection signals (GR3O, GR2O, GR1O (type M)/GR1, GR2(type T). The spindle speed fluctuation detection function does not normally work if the gear count statuses of the two types of signals are different.

CNC system		Serial spindle side		
GR2	GR1	Selected gear	CTH1	CTH2
0	0	1st stage	0	0
0	1	2nd stage	0	1
1	0	3rd stage	1	0
1	1	4th stage	1	1

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (B-64303EN-1)	Path spindle control (T series)
	Spindle speed control
	Spindle output control by the PMC
	Multi-spindle

FANUC SERVO MOTOR αi series MAINTENANCE MANUAL (B-65285EN)
FANUC SERVO AMPLIFIER αi series DESCRIPTIONS (B-65282EN)
FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B-65280EN)

10.19 SPINDLE CONTROL WITH SERVO MOTOR

Overview

The function for spindle control with servo motor allows a servo motor to be used for executing spindle speed commands and spindle functions such as rigid tapping.

(1) Spindle control with servo motor
Velocity control can be performed by using a speed command (S command) with a servo motor set as the tool rotation axis. No reference position return is necessary to switch between rotation and positioning commands.

(2) Spindle indexing
With the spindle indexing function, a stop position can be programmed to stop a rotating axis at the specified position. There are two types of spindle indexing. The first type allows the next-block command to be executed before spindle indexing is finished. The second type allows the next block to be executed only after spindle indexing is completed.
With the first type, it is possible to issue commands to axes other than the axis for which a spindle indexing command has been issued, before the next command is issued to the axis. Before the next command is issued to the axis for which a spindle indexing command has been issued, whether spindle indexing is completed or not can be checked by programming or by using a signal. Using this function can reduce the wait time. In addition, an axis can be stopped at a specified point by issuing a spindle indexing command to the axis when the spindle is rotating.

(3) Axis movement
When bit 0 (PCE) of parameter No. 11006 is 1, if axis movement (G00/G01) is specified for a servo motor spindle in the same way as in a normal controlled axis, position control can be performed.

(4) Rigid tapping with servo motor
Rigid tapping can be carried out by regarding a servo motor spindle as a rotation axis.

(5) Threading, feed per rotation feed, and constant surface speed control
Threading, feed per rotation feed, and constant surface speed control can be performed using a servo motor spindle as a spindle.

(6) Spindle output control with PMC
The rotation speed and polarity can be controlled by PMC.

- Spindle motors and supported functions

Spindle functions	Conventional spindle control	Spindle control with servo motor
Feed per revolution	○	○
Threading	○	○
Polygon machining	○	×*1
Spindle speed fluctuation detection	○	×
Spindle synchronous control	○	×
Simple spindle synchronous control	○	×
Polygon machining with two spindles	○	×
Spindle orientation Multi-point orientation Spindle output switching	○	×
Inter-path spindle control	○	○
Constant surface speed control	○	○
Multi-spindle control	○	○
Rigid tapping	○	○
Spindle output control with PMC	○	○
Actual spindle speed output	○	○
Spindle indexing	×	○

*1 Servo motor can be used as tool rotation axis.
It is not possible to use as a spindle which rotates workpiece.

Notes

(1) This function is optional.

(2) For this function, it is necessary to enable spindle serial output.
For a servo motor spindle, set the type of a spindle motor to a serial spindle. If there is a normal spindle in the path where the servo motor spindle is present, enable multi-spindle control. (For the M series, multi-spindle control is optional function.)

(3) How a servo motor spindle is treated depends on whether position control such as axis movement (G00/G01) is enabled. When bit 0 (PCE) of parameter No. 11006 is 1, position control is enabled. In this case, the servo motor spindle is assumed to be a controlled axis and the number of controlled axes is incremented.
When bit 0 (PCE) of parameter No. 11006 is 0, position control is disabled. In this case, the servo motor spindle is not assumed to be a controlled axis and the number of controlled axes is not incremented.

Maximum number of controlled axes

Maximum number of controlled axes in an entire system	Position control	Controlled axis type	Maximum number of servo motors
M series / 1-path system Max. number of feed axes 7axes Max. number of spindle axes 2axes Max. total number of control axes 8axes	Disabled	Servo axis + Servo motor spindle	8
		Only servo axis	7
	Enabled	Servo axis + Servo motor spindle	7
		Only servo axis	7
T series / 1-path system Max. number of feed axes 7axes Max. number of spindle axes 3axes Max. total number of control axes 8axes	Disabled	Servo axis + Servo motor spindle	8
		Only servo axis	7
	Enabled	Servo axis + Servo motor spindle	7
		Only servo axis	7
T series / 2-path system Max. number of feed axes 9axes Max. number of spindle axes 4axes Max. total number of control axes 11axes	Disabled	Servo axis + Servo motor spindle	9 (Note*)
		Only servo axis	9
	Enabled	Servo axis + Servo motor spindle	9
		Only servo axis	9

(Additional explanation 1)
"number of feed axes + number of spindle axes" is defined "total number of control axes"

(Additional explanation 2)
The Max. number of axes for each path in the T series / 2-path system is the same as T series / 1-path system.

(Note*) The number of connectable servo motors is up to 9.

(4) A servo motor spindle used by this function is counted as a controlled spindle.

Maximum number of spindles

Maximum number of spindles in an entire system	Spindle type	Maximum number of spindles
M series / 1-path system 2 axes	Spindle + Servo motor spindle	2
	Only spindle	2
T series / 1-path system 3 axes	Spindle + Servo motor spindle	3
	Only spindle	3
T series / 2-path system 4 axes	Spindle + Servo motor spindle	4
	Only spindle	4

(5) The servo motor spindle can be set up to the maximum number of spindles. But, in case of setting that the total number of the feed axes and the spindle axes exceeds the Max. total number of control axes, alarm PS0365 is issued and the emergency stop state is entered.

(6) When performing rigid tapping with a servo motor, enable rigid tapping (bit 3 (NRG) of parameter No. 8135 is 0).
(7) When bit 0 (PCE) of parameter No. 11006 is 1, position control is enabled and a servo motor spindle can be set as a PMC controlled axis.
If a servo motor spindle is set as a PMC controlled axis when position control is disabled (bit 0 (PCE) of parameter No. 11006 is 0), alarm DS2003 is issued.
(8) This function cannot be used in system composed of servo motor spindle in serial/analog spindle control and an analog spindle.
The serial spindle is also necessary for above system.

10.19.1 Spindle Control with Servo Motor

Specification

- Command with a program

This function can be used to specify the S spindle rotation command for a servo motor spindle in the SV speed control mode. For rotation control of a servo motor, specify G96.4 to start the SV speed control mode. Once the SV speed control mode is specified, the S command is valid for the servo motor until the SV speed control mode is released. Do not specify a positioning command in the SV speed control mode. Otherwise, alarm PS0445 is issued.
Before positioning operation, release the SV speed control mode. To release the SV speed control mode, specify the spindle indexing command G96.1/G96.2. For details on spindle indexing, see "Spindle Indexing Function". When bit 0 (PCE) of parameter No. 11006 is 1, a positioning command can be specified. When bit 0 (PCE) of parameter No. 11006 is 0, a positioning command cannot be specified for a servo motor spindle. If specified, alarm PS0601 is issued.
In a mode other than the SV speed control mode, the S command is invalid. However, since information of the S command is recorded, if the SV speed control mode is entered, rotation starts at the specified rotation speed.

- Command with a signal

SV speed control mode signal <Gn521> can also be used to start and cancel the SV speed control mode. The SV speed control mode is started or canceled on a rising or falling edge of the SV speed control mode signal. Therefore, to start the SV speed control mode again after the SV speed control mode is started by using the signal then canceled by a programmed command, input the SV speed control mode signal again or specify G96.4.
The SV speed control mode status can be checked using the SV speed control mode in-progress signals (SVREV1 to SVREV5 <Fn521.0 to Fn521.4>).
When the signal is set to 0 during rotation, spindle indexing is performed, then the SV speed control mode is canceled. Spindle indexing performs positioning to R0 (absolute value 0). For spindle indexing, see "Spindle Indexing Function".

Format

G96.4 P_ ; **Start SV speed control mode**

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

- Starting the SV speed control mode

When multi-spindle control is enabled and a spindle is selected with address P (Bit 3 (MPP) of parameter No. 3703 = "1"), the SV speed control mode can be started by specifying G96.4 and spindle selection command P. To specify address P, use parameter No. 3781 (P code for selecting a spindle). When a spindle is selected with spindle selection signals SWS1 to SWS2 <Gn027.0 to Gn027.1>, it is possible to start the SV speed control mode if spindle selection signal is input when program is analyzed in the G96.4 command block. In any case, when a servo motor spindle has not been selected in the G96.4 command block, alarm PS0602 is issued.
When multi-spindle control is disabled, it is possible to start the SV speed control mode if G96.4 is independently commanded. When there is no servo motor spindle in the system, alarm PS0602 is issued.

- Canceling the SV speed control mode

When multi-spindle control is enabled and a spindle is selected with address P (Bit 3 (MPP) of parameter No. 3703 = "1"), the SV speed control mode can be canceled by specifying a spindle indexing command (G96.1/G96.2) and spindle selection command P. When a spindle is selected with spindle selection signals SWS1 to SWS2 <Gn027.0 to Gn027.1>, it is possible to cancel the SV speed control mode if spindle selection signal is input when program is analyzed in the spindle indexing command (G96.1/G96.2) block. In any case, when a servo motor spindle has not been selected, alarm PS0602 is issued.
When multi-spindle control is disabled, it is possible to cancel the SV speed control mode if G96.4 is independently commanded. When there is no servo motor spindle in the system, alarm PS0602 is issued.
For the command format of the spindle indexing command, see "Spindle Indexing Function".

- Notes

To specify “G96.4 P_ ;”, use an independent block.
However, it is possible to specify "G96.4 P_ S_ ;".
Moreover, in 2-path system, when it is specified from paths other than the path that a servo motor spindle belongs, alarm PS0602 is issued.

Explanation

- Command

(1) Spindle speed command output
Set up the spindle speed command in the same way as for the ordinary speed command (S command). Before specifying a speed command (S command), start the SV speed control mode. When performing positioning, cancel the SV speed control mode, and select the position control mode.
The sequence by the following input signals is unnecessary.
*ESPA, MRDYA, and SFRA

(2) Condition for stopping spindle speed output
Commands once output to a spindle become 0 if *SSTP becomes “0” or if a command (such as S0) that makes the spindle speed command output 0 is issued. Also, issuing a spindle indexing command (G96.1/G96.2) makes the spindle speed command output 0. In addition, an emergency stop condition and servo alarm bring the spindle to a stop.
With M05, the CNC does not make the command output to the spindle 0.

(3) Stopping rotation by spindle indexing
Issuing a command that specifies a position enables the rotating axis to stop at the specified position.
For details, see Subsection, "Spindle Indexing Function."

(4) Maximum speed
The maximum speed that can be specified is usually 2777 min^{-1}.
However, setting bit 3 (IRC) of parameter No. 1408 to "1" may be able to increase the limit to about 27770 min^{-1} depending on the performance of the related motor and detector.

- Multi-spindle control and commands from another path

When a servo motor spindle and another spindle are present together in the same path, the multi-spindle control function is required to specify a rotation command. The path spindle control function can handle commands from another path. The address P-based spindle select function for multi-spindle control can be used to select spindles in another path.
Example program commands follow.
Example 1: (Spindle selection with address P)
Bit 3 (MPP) of parameter No. 3703 = "1": A spindle is selected with address P.

Spindle configuration (S1 = first spindle and S2 = second spindle)

Path 1	Path 2
S1 (spindle axis)	S3 (spindle axis)
S2 (servo motor spindle)	-

Setting address P for spindle selection in multi-spindle control

Parameter	Path 1	Path 2
3781	11 (S1)	21 (S3)
	12 (S2)	-

Program example

Command issuing path	Program	Operation
1	M03 S1000 P12 ;	S2 rotates in normal direction at 1000 min^{-1}.
1	M03 S1500 P21 ;	S3 rotates in normal direction at 1500 min^{-1}.
2	M04 S1500 P11 ;	S1 rotates in reverse direction at 1500 min^{-1}.

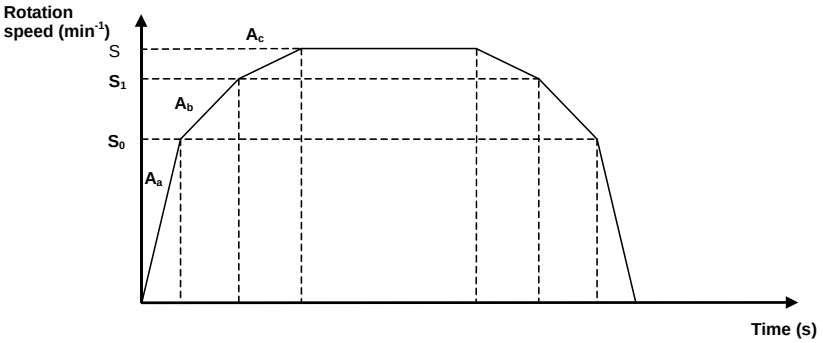
- Manual operation of a servo motor spindle

Before performing the manual operation and manual reference position return of a servo motor spindle, release the SV speed control mode. Manual operation and manual reference position return in the SV speed control mode are invalid.
When using an absolute position detector, manual reference position return is not required. If reference position return (G28) is performed in a program when position control is disabled (bit 0 (PCE) of parameter No. 11006 is 0), alarm PS0601 is issued.

NOTE
If a movement command is specified for a servo motor spindle when position control is disabled (bit 0 (PCE) of parameter No. 11006 is 0), alarm PS0601 is issued.

- Acceleration/deceleration (time constant)

It is possible to change the spindle acceleration/deceleration specified in a rotation command according to the speed of the spindle.
Acceleration/deceleration can be switched at two points of speed, using parameters S0 and S1 (switching speed). In addition, parameters Aa, Ab, and Ac are available to set up three acceleration/deceleration spans.



S_0 : Setting of parameter No. 11020 (acceleration/deceleration is switched at rotation speed S_0 (min^{-1})).
S_1 : Setting of parameter No. 11021 (acceleration/deceleration is switched at rotation speed S_1 (min^{-1})).
S : Specified rotation speed (min^{-1}).
Aa : Setting of parameter No. 11030 (acceleration/deceleration (min^{-1}/s) used between rotation speeds 0 and S_0).
Ab : Setting of parameter No. 11031 (acceleration/deceleration (min^{-1}/s) used between rotation speeds S_0 and S_1).
Ac : Setting of parameter No. 11032 (acceleration/deceleration (min^{-1}/s) used between S_1 and the specified speed S).

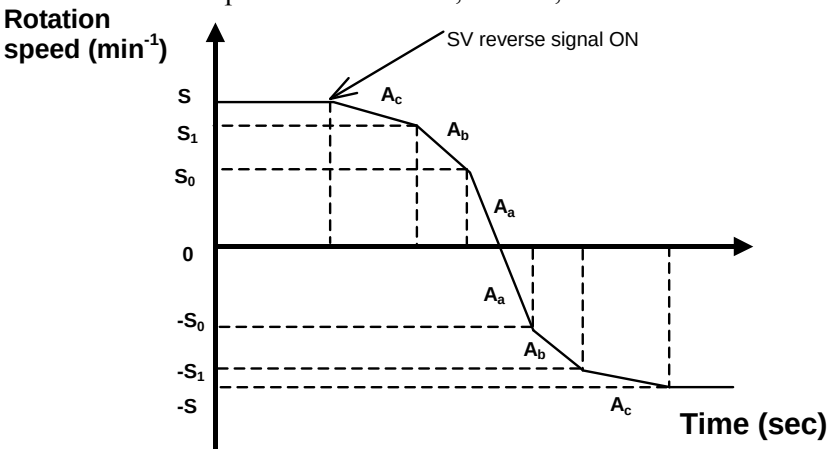
Determine the setting of each parameter according to the torque characteristic of the motor.

- Acceleration/deceleration after interpolation

Acceleration/deceleration after interpolation is available in the SV speed control mode. Bit 1 (TCR) of parameter No.11001 can be used to select a time constant type, that is, parameter No. 1622
(Time constant of acceleration/deceleration in cutting feed for each axis) or parameter No. 11016 (Time constant of acceleration/ deceleration in SV speed control mode for each axis).

- Direction of rotation

The voltage polarity at the time of spindle speed output can be changed using bit 6 (CWM) and bit 7 (TCM) of parameter No. 3706.
It is also possible to reverse the direction of spindle rotation, using the SV reverse signal <Gn523>.
These functions are usable in the rotation mode and during rigid tapping. Changing the signal to ON/OFF during rotation causes the spindle to decelerate, reverse, and then accelerate.



- Display

Bit 3 (NDF) of parameter No. 3115 can be used to specify whether to display the actual speed. This, however, is not considered in the SV speed control mode regardless of the setting of the NDF parameter bit. In addition, bits 0 (NDP) and 1 (NDA) of parameter No. 3115 can be used to specify whether to display, respectively, the current position and the remaining amount of movement.

- Setting a servo motor spindle

(1) Axis setting

Set the servo axis number (parameter No. 1023) of an axis used as a servo spindle to a value as a servo motor, as usual. Bit 7 (SRV) of parameter No. 11000 sets the servo motor used as a servo motor spindle. Parameter No. 11010 sets the spindle used as the servo motor spindle by a spindle number. In this case, set the spindle amplifier number (parameter No. 3717) of the set spindle to 0. If the spindle amplifier number is not 0, alarm PW1111 is issued.

In multi-spindle control, set bit 3 (MPP) of parameter No. 3703 to "1" to specify that program commands be used to select a spindle.

Also set parameter No. 3781 with the P code for selecting a spindle.

(2) Using a command to specify a spindle speed requires adjusting the following parameters.

Parameter No.11013: Positioning deviation limit in movement
Parameter No. 11014: Positioning deviation limit in the stop state
Parameter No. 11015: Maximum motor speed

(3) Rotation axis setting

Using bits 0 (ROTx) and 1 (ROSx) of parameter No. 1006, select rotation axis type A.

Using the rollover function, round the absolute coordinate values to be displayed to an amount of movement per rotation specified in parameter No.1260 in order to prevent rotation axis coordinate overflow. Also round the relative coordinate values to be displayed to an amount of movement per rotation by setting bit 2 (RRLx) of parameter No. 1008 to "1".

The rollover function is enabled by setting bit 0 (ROAx) of parameter No. 1008 to "1" (for the rotation axis).

- Example of parameter setting

The following examples show typical parameter settings. These typical parameter settings should be used unless otherwise stated.

(1) Spindle setting

Let the fourth axis of the CNC be the C axis. Assuming that the fourth axis is connected to the second spindle, let us use the second spindle as a servo motor axis. If the amplifier number (parameter No. 3717) of the second spindle is not 0, alarm PW1111 is issued.

No.11000#7 (C) = "1"
No.11010 (C) = 2 (a spindle number is specified for a controlled axis assigned to the servo motor spindle.)
(Set up No. 3717 (S2) = 0.)

Given below are the least command increment, detection unit, and the amount of movement per rotation for the C axis.

$$\text{Least command increment} = \frac{L \times CMR}{\mathrm{Q} \times DMR}$$

$$\text{Detection unit} = \frac{\text{Least command increment}}{CMR} = \frac{L}{\mathrm{Q} \times DMR}$$

$$\text{Amount of movement per tool axis rotation} = \frac{360}{\text{Least command increment}}$$

where

L: Amount of tool movement per motor rotation
(360 × speed increment/decrement ratio) [deg]

Speed increment/decrement ratio:
Ratio of maximum spindle speed (No.3741 to 3744) corresponding to gear under selection to maximum motor speed(No.11015)
= (No.3741 to 3744) / No.11015

For example, if the servo motor is connected directly to the spindle (live tool), L = 360. For example, if the speed must be increased by twice, L = 720. However, when it is necessary to consider the gear ratio, assume L=360 and consider it with flexible feed gear (above DMR).

Q: Number of pulses per pulse coder rotation
(For the serial pulse coder, Q = 1000000.)

The least command increment mentioned above is specific to the C axis and is determined independently of bits 0 (IS-A) and 1 (IS-C) of parameter No 1013.
Reset both bits 0 (IS-A) and (IS-C) of parameter No 1013 to "0" to select IS-B.
If the servo motor is connected directly to the spindle (live tool):

$$\text{Least command increment} = \frac{360 \times 1}{1000000 \times 36/100} = 0.001 \quad [\text{deg}]$$

Detection unit = 0.001 [deg]
Amount of movement per tool axis rotation =360.0 [deg]

So, the required parameter setting must be:

Parameter No. 11011 (C) = 360.000 (amount of movement per servo motor spindle rotation)

In addition, the acceleration/deceleration values for rotation control must be specified in parameter Nos. 11020 to 11032.

Let us assume that the axis parameters be specified for the fourth axis, as follows:

(2) Servo parameter setting
[When the gear ratio of the tool axis to the motor is 1:1]
Let us assume: CMR = 1 and DMR = 36/100
(The capacity of the reference counter will be 360000.)

No.1820 (C) = 2 (CMR)
No.1821 (C) = 360000 (Reference counter capacity)
No.2084 (C) = 36 (DMR numerator)
No.2085 (C) = 100 (DMR denominator)
No.11015(C) = 5000 (Maximum motor speed)
No.3741(S2) = 5000 (Maximum speed for gear 1)

[When the gear ratio of the tool axis to the motor is 2:1]
DMR is changed as follows.

No.2084 (C) = 72 (DMR numerator)
No.2085 (C) = 100 (DMR denominator)

For the other servo parameters, use the respective typical settings.

Restrictions

(1) This function is optional.
(2) For this function, it is necessary to enable spindle serial output (set bit 5 (SSN) of parameter No.8133) to 0).
Be sure to set the spindle motor type of servo motor spindle as a serial spindle (bit 0 (A/S) of parameter No.3716 = "1").
If there is a normal spindle in the path where the servo motor spindle is present, enable multi-spindle control (set bit 3 (MSP) of parameter No.8133) to 1). (For the M series, multi-spindle control is optional function.)
(3) This function assumes a servo motor spindle to be a spindle.
(4) For this function, it is necessary to select T type gear (Bit 4 (GTT) of parameter No. 3706 = "1"). (M series)
(5) When multi-spindle control is enabled, in settings other than multi-spindle P type, specifying G96.* P is regarded as invalid.
(6) When starting or canceling the SV speed control mode during automatic operation, be sure to specify the path of the target axis of the SV speed control mode. When it is specified from the other path, alarm PS0602 is issued.

(7) Be sure to set M code which starts or cancels the SV speed control mode to M code preventing buffering (parameter No.3411 to 3420, No.3421 to 3432).
(8) When the SV speed control mode is started or cancelled except for automatic operation, be sure to perform manual reference position return before starting the automatic operation so that the target axis lose its reference position.
The axis may not move correctly to the commanded position in SV speed control cancel mode (position control mode) when the automatic operation is started without performing manual reference position return.

Notes

(1) Unlike spindle motors, a servo motor spindle stops rotating when a servo alarm, an emergency stop, or a machine lock occurs.
(2) The maximum motor speed is obtained by applying the feed gear to the maximum motor speed set in parameter No. 11015.
(3) If bit 3 (IRC) of parameter No. 1408 is set to 0, the error prevention function may work when the speed has reached around 2778 min^{-1}. When a speed of 2778 min^{-1} or more is required, set the IRC parameter bit to 1. If IRC is set to 1, it becomes possible to specify a speed of up to approximately 27770 min^{-1}.

Signal

SV speed control mode signals SRVON1 to SRVON6 <Gn521.0 to Gn521.5>

[Classification] Input signal
[Function] These signal switch the respective axes to the SV speed control mode.
[Operation] Setting these signals to "1" cause the respective axes to enter the SV speed control mode.
Resetting these signals to "0" cause the respective axes to exit the SV speed control mode.
Resetting the signals to 0 during axis rotation causes the respective axes to start spindle indexing, to stop at the origin (R0), and to exit the SV rotation mode.

NOTE
Even if this signal is set to 1, when the SV speed control mode is canceled (the SV speed control mode in-progress signal Fn521 is set to 0) by a programmed command, the spindle speed control function is disabled. To enable the function, set this signal again.

SV reverse signals SVRVS1 to SVRVS6 <Gn523.0 to Gn523.5>

[Classification] Input signal
[Function] These signals cause the respective axes to reverse their rotation direction in the SV speed control mode.
[Operation] Setting these signals to "1" cause the respective axes to reverse their rotation direction.
Resetting these signals to "0" cause the respective axes to resume their normal rotation direction.

NOTE
These signals are effective even in rigid tapping.

SV speed control mode in-progress signals SVREV1 to SVREV6 <Fn521.0 to Fn521.5>

[Classification] Output signal
[Function] These signals inform that the respective axes are in the SV speed control mode.
[Output cond.] These signals are "1" if:
- The respective axes are in the SV speed control mode.

These signals are "0" if:

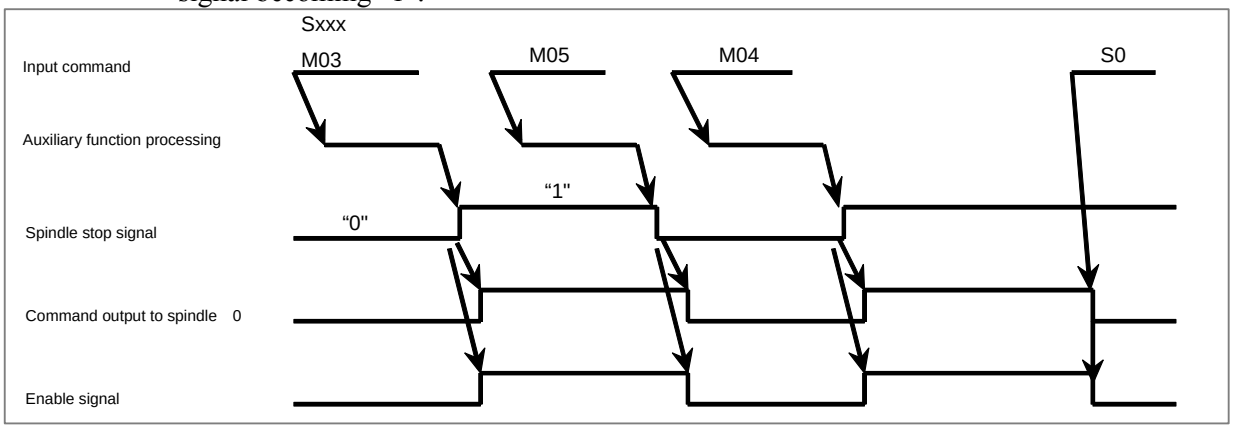
- The respective axes are in the SV speed control mode, or
- The respective axes are in the position control mode.

Spindle stop signal *SSTP<Gn029.6>

[Classification] Input signal

[Function] This signal stops the command output to the spindle.

[Operation] When this signal becomes "0", the command output to the spindle becomes 0, resulting in the enable signal becoming "0". So, M05 is not output. When this signal becomes "1", the command output to the spindle resumes its previous value, resulting in the enable signal becoming "1".



The timing chart above is illustrative only. To be more precise, follow a timing chart that meets the specification of the speed control unit for the spindle motor in use.

- Keep this signal set to "1" unless it is used.
- The CNC uses the commands M03, M04, and M05 to send only code and strobe signals.

Individual spindle stop signals *SSTP1<Gn027.3>, *SSTP2<Gn027.4>

[Classification] Input signal

[Function] These signals are usable only during multi-spindle control.
They cause the respective spindles to stop.

*SSTP1 1: Does not make the output to the first spindle 0 min^{-1}.
0: Makes the output to the first spindle 0 min^{-1}.

*SSTP2 1: Does not make the output to the second spindle 0 min^{-1}.
0: Makes the output to the second spindle 0 min^{-1}.

First spindle speed override signals SOV0 to SOV7<Gn030>
Second spindle speed override signals SOV20 to SOV27<Gn376>

[Classification] Input signal

[Function] When bit 4 (EOV) of parameter No. 3713 is set to 1, an override of 0 to 254% can be applied to the speed directed for each axis to the CNC.

[Operation] Set an override value with an 8-bit binary value.
If all of the bits are set to 1, however, an override of 0% is assumed.

- Tapping cycle (for the T series, G84 and G88)
- Threading mode (for the T series, G32, G92, and G76)

→ Specify an override of 100% unless this function is used.
(An override of 0% is applied, preventing spindle rotation.)

NOTE
When bit 4 (EOV) of parameter No. 3713 is set to 0, SOV0 to SOV7 are used as override signals for all spindles for which a speed is specified.
SOV20 to SOV27 are ignored.

Spindle enable signals ENB<Fn001.4>, ENB2<Fn038.2>

[Classification] Output signal

[Function] These signals inform whether there is an output to the second spindle if the multi-spindle control function is available.
The signals can be used as a condition for stopping the analog spindle and in PMC ladder sequences related to rigid tapping.
(See Subsection, "Rigid Tapping.")

[Output cond.] ENB"1" if the output to the first spindle control unit is not 0.
"0" if the output to the first spindle control unit is 0.
ENB2 "1" if the output to the second spindle control unit is not 0.
"0" if the output to the second spindle control unit is 0.

Address P signals MSP00 to MSP15<Fn160.0 to Fn161.7>

[Classification] Output signal

[Function] These signals output the P value most recently specified by the S_ P_ ; command.

[Output cond.] If the address P-based multi-spindle control function is enabled (bit 3 (MPP) of parameter No. 3703 ="1"), the P value specified by the S_ P_ ; command is output. If no S_ P_ ; has been issued since application of the power, the initial P value specified in parameter No. 3775 is output.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Gn027				*SSTP2	*SSTP1			
Gn029		*SSTP						
Gn030	SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0
Gn376	SOV27	SOV26	SOV25	SOV24	SOV23	SOV22	SOV21	SOV20
Gn521			SRVON6	SRVON5	SRVON4	SRVON3	SRVON2	SRVON1
Gn523			SVRVS6	SVRVS5	SVRVS4	SVRVS3	SVRVS2	SVRVS1
Fn001				ENB				
Fn038						ENB2		
Fn160	MSP07	MSP06	MSP05	MSP04	MSP03	MSP02	MSP01	MSP00
Fn161	MSP15	MSP14	MSP13	MSP12	MSP11	MSP10	MSP09	MSP08
Fn521			SVREV6	SVREV5	SVREV4	SVREV3	SVREV2	SVREV1

Parameter

The major related parameters are described below.

0981	Absolute path number to which each axis belongs

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] 1, 2
Set the path to which each axis belongs.

NOTE
1 When 0 is set for all axes, the parameter is automatically set according to the number of controlled axes of each path.
2 When the setting falls outside the range, the axis is assumed to belong to the first path.

0982	Absolute path number to which each spindle belongs

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] 1, 2
Set the path to which each spindle belongs.

NOTE
1 When 0 is set for all axes, the parameter is automatically set according to the number of controlled axes of each path.
2 When the setting falls outside the range, the axis is assumed to belong to the first path.
3 When spindle control with servo motor is enabled, the servo motor used as the spindle controlled axis is treated as a spindle. Therefore, it is necessary to set the path to which the axis subject to spindle control with servo motor.

	#7	#6	#5	#4	#3	#2	#1	#0
1006							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 rotation axis.

#1 ROSx

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	Rotation axis (A type) (1) Inch/metric conversion is not done. Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values are rounded or not rounded by parameter No.1008#0(ROAx) and #2(RRLx). (2) Stored pitch error compensation is the rotation type. (Refer to parameter No.3624) (3) 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	Rotation axis (B type) (1) Inch/metric conversion, absolute coordinate values and relative coordinate values are 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 rotation axis roll over function and the index table indexing function (M series)
Except for the above.		Setting is invalid (unused)

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	Rotation 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 Number of controlled axes
Set the servo axis for each control axis.
Usually set to same number as the control axis number.
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, ...) 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, ...) 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.

1260	The shift amount per one rotation of a rotation 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 rotation axis.
For the rotation axis used for cylindrical interpolation, set the standard value.

	#7	#6	#5	#4	#3	#2	#1	#0
1408					IRCx			

[Input type] Parameter input
[Data type] Bit axis

#3 IRCx The least input increment of the maximum cutting feedrates set in parameter Nos. 1430 and 1432 is:
0: Not multiplied by ten.
1: Multiplied by ten.
Set this parameter for the following axes, which are operated by the following functions:

- Servo motor spindle

- Tool rotation axis in the polygon turning function

To a rotation speed of 1000 (1/min) (=360000 (deg/min)) is to be used when this parameter is set to 1, set 36000.0 in parameter No. 1430/1432.

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.

1825	Servo loop gain for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] 0.01/sec
[Valid data range] 1 to 9999
Set the loop gain for position control for each axis.
When the machine performs linear and circular interpolation (cutting), the same value must be set for all axes. When the machine requires positioning only, the values set for the axes may differ from one another. As the loop gain increases, the response by position control is improved. A too large loop gain, however, makes the servo system unstable.
The relationship between the positioning deviation (the number of pulses counted by the error counter) and the feedrate is expressed as follows:

Positioning deviation = Feedrate / (60 × Loop gain)

Unit : Positioning deviation mm, inch or deg
Feedrate mm/min, inch/min, or deg/min
Loop gain 1/sec

1826	In-position width for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999
The in-position width is set for each axis.

When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in-position width, the machine is assumed to have reached the specified position. (The machine is in the in-position state.)

1827	In-position width in cutting feed for each axis

[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 each axis in cutting feed. This parameter is used when bit 4 (CCI) of parameter No.1801=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) 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) is generated, and operation is stopped immediately (as in emergency stop).

3411	M code preventing buffering 1
3412	M code preventing buffering 2
:	
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.

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.

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 parameter 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
1 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.
2 If the minimum value is greater than the maximum value, the setting is invalid.
3 If there is only one data item, the minimum value must be equal to the maximum value.

	#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.

NOTE
Make the setting on the side of the path in which multi-spindle control is unnecessary in 2-path control.

	#7	#6	#5	#4	#3	#2	#1	#0
3703					MPP			2P2
					MPP			

[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 2P2 When a 2-path system is used, inter-path spindle control allows:
0: Configuration where the spindle that belongs to one path only is shared between path 1 and path 2.
1: Configuration where the spindles that belong to path 1 and 2 are shared between the two paths.
When the spindle that belongs to an arbitrary path is shared between arbitrary paths, set bit 2 (MPM) of parameter No.3703. (The meanings of signals used vary, so that ladder program modifications need to be made.)

#3 MPP In multi-spindle control, a spindle selection using a programmed command instead of using the signals (SWS1 to SWS2<G027.0, 1>) is:
0: Not made.
1: Made.

NOTE
When this parameter is set to 1, set parameter No. 3781 at the same time.

	#7	#6	#5	#4	#3	#2	#1	#0
3706	TCW	CWM						
	TCW	CWM		GTT				

[Input type] Parameter input
[Data type] Bit path

#4 GTT Selection of a spindle gear selection method
0: Type M.
1: Type T.

NOTE
1 M type
The gear selection signal is not input. The CNC selects a gear based on the speed range of each gear set by a parameter beforehand according to S codes, and the selected gear is posted by outputting the gear selection signal. Moreover, the spindle speed matching the gear selected by the output gear selection signal is output.
T type
The gear selection signal is input. The spindle speed matching the gear selected by this signal is output.
2 When constant surface speed control is selected (bit 0 (SSC) of parameter No. 8133 is 1), the T type is assumed regardless of the setting of this parameter.
3 When type T spindle gear switching is selected, the following parameters have no effect:
No.3705#2(SGB), No.3751, No.3752,
No.3705#1(GST), No.3705#3(SGT), No.3761, No.3762,
No.3705#6(SFA), No.3735, No.3736
On the other hand, parameter No. 3744 becomes usable.

#6 CWM
#7 TCW The polarity of the spindle speed output voltage is specified as listed below.

TCW	CWM	Voltage polarity
0	0	Both M03 and M04 positive
0	1	Both M03 and M04 negative
1	0	M03 positive, M04 negative
1	1	M03 negative, M04 positive

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.
3: Spindle motor connected to amplifier number 3 is used.

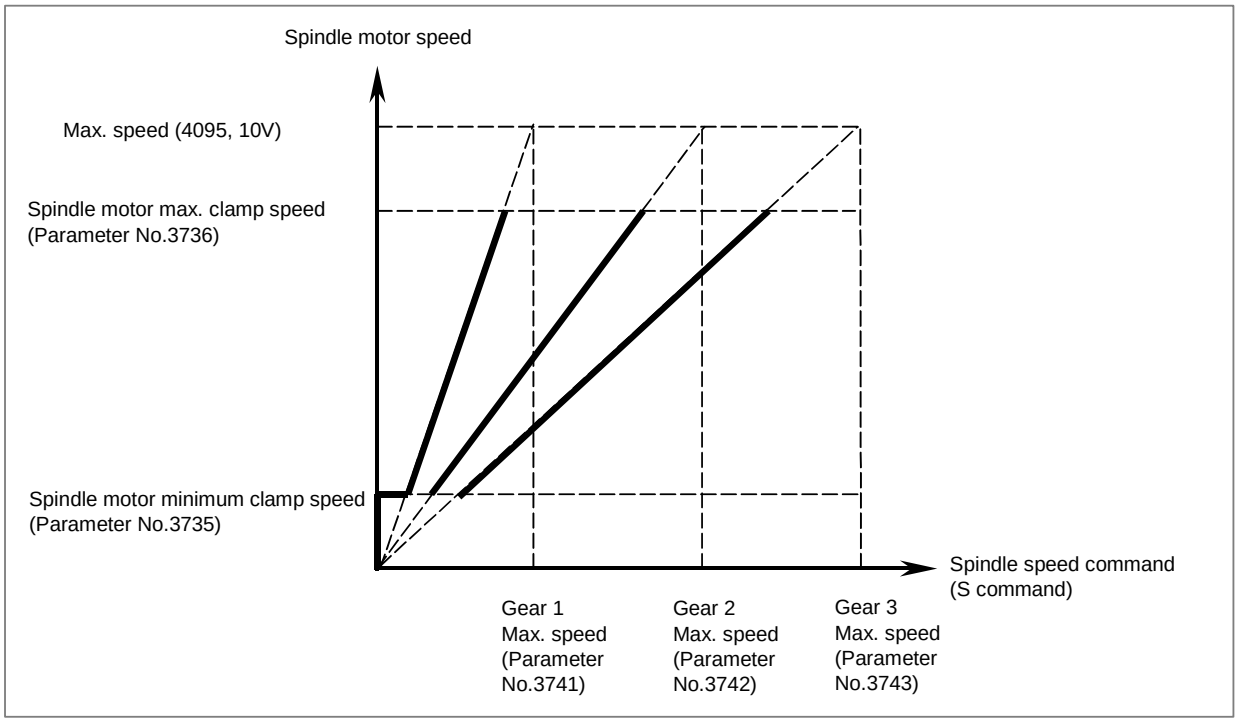
NOTE
When using an analog spindle, set it at the end of the spindle configuration.
(Example)
When there are three spindles in an entire system (two serial spindles and one analog spindle), set the spindle amplifier number (this parameter) of the analog spindle to 3.

3718	Subscript for display of a serial spindle or analog spindle

[Input type] Parameter input
[Data type] Byte spindle
[Valid data range] 0 to 122
Set a subscript to be added to spindle speed display on a screen such as the position display screen.

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.



3772	Maximum spindle speed

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999
This parameter sets the maximum spindle speed.
When a command specifying a speed exceeding the maximum speed of the spindle is specified , or the speed of the spindle exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

⚠ CAUTION
1 When 0 is set in this parameter, the speed of the spindle is not clamped.
2 When spindle speed command control is applied using the PMC, this parameter has no effect, and the spindle speed is not clamped.

3775	Default P command value for spindle selection in multi-spindle control

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 32767
When bit 3 (MPP) of parameter No. 3703 is set to 1 and bit 2 (MPA) of parameter No. 3706 is set to 1 in multi-spindle control, set a default P command value applicable if S_P_ is not specified even once after power-up.

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.

NOTE
1 This parameter is valid if bit 3 (MPP) of parameter No. 3703 is set to 1.
2 If this parameter is set to 0, the corresponding spindle cannot be selected by a P code.

NOTE
3 Under 2-path control, the P code specified here is valid for each path.
For instance, if the P code to select the first spindle of path 2 is set to 21, specifying S1000 P21; in path 1 causes the first spindle of path 2 to be rotated at S1000.
4 Identical P code values cannot be used for different spindles. (Identical P code values cannot be used even if the paths are different.)
5 When this parameter is used (when bit 3 (MPP) of parameter No. 3703 is set to 1), the spindle command selection signal is invalid.
6 To use this parameter, enable multi-spindle control (bit 3 (MSP) of parameter No. 8133 is 1).

8130	Number of controlled 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] 1 to Maximum number of controlled axes
This parameter sets the number of axes for each path.

NOTE
When spindle control with servo motor is enabled, set the number of axes including this axis for the axes with a spindle controlled axis with servo motor.

	#7	#6	#5	#4	#3	#2	#1	#0
11000	SRVx			PCAx				SPCx

[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 SPCx SV speed control mode of spindle control with servo motor is:
0: Position control type.
1: Velocity control type.

#4 PCAx For parameter PCE(No.11006#0)=0, to each axis, positional control under spindle control with servo motor is:
0: Disabled.
1: Enabled.

#7 SRVx In spindle control with servo motor:
0: Use as the servo motor spindle is not performed.
1: Use as the servo motor spindle is performed.

NOTE
For the axes that use spindle control with servo motor, set also parameter No. 11010.

	#7	#6	#5	#4	#3	#2	#1	#0
11001							TCRx	

[Input type] Parameter input
[Data type] Bit axis

#1 TCRx When the speed is controlled under spindle control with servo motor, acceleration/deceleration after interpolation:
0: Enables parameter No. 1622 (time constant of cutting feed acceleration/deceleration for each axis)
1: Enables parameter No.11016 (time constant specific to speed control)
Set the parameter for the axis subject to spindle control with servo motor.

	#7	#6	#5	#4	#3	#2	#1	#0
11006								PCE

[Input type] Parameter input
[Data type] Bit

NOTE
When this parameter is set, the power must be turned off before operation is continued.

#0 PCE Positional control under spindle control with servo motor is:
0: Disabled.
1: Enabled.

NOTE
This parameter enables positional control with all axes with spindle control with servo motor. To each axis, when you want to set positional control enabled or disabled, set to parameter PCA(No.11000#4).

11010	Number of an axis subject to spindle control with servo motor

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 maximum number of controlled spindles
This parameter sets a spindle number for a servo axis subject to spindle control with servo motor.

NOTE
Set a spindle number for the axis set by bit 7 of parameter No. 1100. Set 0 for an axis not subject to spindle control with servo motor.

11011	Movement of servo motor spindle per revolution

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)
This parameter sets the amount of movement per revolution of the servo motor spindle in spindle control with servo motor.

11012	Spindle indexing speed for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] min^{-1}
Set the spindle indexing speed applied when the function for spindle control with servo motor is performed.
If 0 is set, the value set in parameter No. 11020 (acceleration/deceleration switching speed (S_0) for each axis) is assumed to be the spindle indexing speed.

11013	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
This parameter sets the positioning deviation limit during movement for each axis in spindle control with servo motor.

11014	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
This parameter sets the positioning deviation limit during a stop for each axis in spindle control with servo motor.

11015	Maximum motor speed

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}

[Valid data range] 0 to 99999999
This parameter sets the maximum motor speed in spindle control with servo motor.

11016	Time constant of acceleration/deceleration in SV speed control mode for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
This parameter sets acceleration/deceleration after interpolation during speed control in spindle control with servo motor. Set it for the axis of a servo motor spindle. 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. Type of acceleration/deceleration is applied by parameter CTLx, CTBx (No.1610#0, #1).

11017	FL rate of exponential acceleration/deceleration in SV speed control mode 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 the lower limit speed (FL speed) of exponential acceleration/deceleration during speed control in spindle control with servo motor.
Set it for the axis of a servo motor spindle.

11020	Acceleration/deceleration switching speed (1st) for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999
This parameter sets the speed at which acceleration/deceleration is switched during rotation control in spindle control with servo motor. (First step)

11021	Acceleration/deceleration switching speed (2nd) for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999
This parameter sets the speed at which acceleration/deceleration is switched during rotation control in spindle control with servo motor. (Second step)

11030	Individual acceleration / deceleration 1 (Leg 1)

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}/s
[Valid data range] 0 to 100000

This parameter sets the acceleration/deceleration used during rotation control in spindle control with servo motor. When the speed ranges from 0 to acceleration switching speed 1, acceleration/deceleration 1 is applied. Acceleration switching speed 1 is the speed set in parameter No. 11020.

11031	Individual acceleration / deceleration 2 (Leg 2)

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}/s
[Valid data range] 0 to 100000

This parameter sets the acceleration/deceleration used during rotation control in spindle control with servo motor. When the speed ranges from acceleration switching speed 1 to acceleration switching speed 2, acceleration/ deceleration 2 is applied. Acceleration switching speed 1 and acceleration switching speed 2 are the speeds set in parameter Nos. 11020 and 11021, respectively.

11032	Individual acceleration / deceleration 3 (Leg 3)

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}/s
[Valid data range] 0 to 100000

This parameter sets the acceleration/deceleration used during rotation control in spindle control with servo motor. The speed is acceleration/deceleration 3 in the range from acceleration switching speed 2 to the maximum speed. Acceleration switching speed 2 is the speed set in parameter No. 11021.

Alarm and message

Number	Message	Description
PS0365	TOO MANY MAXIMUM SV/SP AXIS NUMBER PER PATH	The Max. total number of control axes, the Max. number of feed axes or the Max. number of control axes is exceeded. Check parameter No. 0981 and No. 0982. If this alarm is generated, the emergency stop state cannot be released.
PS0445	ILLEGAL AXIS OPERATION	A positioning command was specified in the rotation control mode. Check the SV speed control mode in-progress signal.
PS0446	ILLEGAL COMMAND IN G96.1/G96.2/G96.3/G96.4	G96.1, G96.2, G96.3, or G96.4 is specified with another command in the same block. Correct the program.
PS0447	ILLEGAL SETTING DATA	The setting of an axis subject to spindle control with servo motor is illegal. Check the parameters for the function for spindle control with servo motor.
PS0601	ILLEGAL AXIS OPERATION FOR SERVO MOTOR SPINDLE	A movement command was specified for an axis subject to spindle control with servo motor. Correct the program.
PS0602	ILLEGAL AXIS OPERATION FOR LIVE TOOL AXIS	An axis subject to spindle control with servo motor is not correctly selected.

Number	Message	Description
PW1110	ILLEGAL PARAMETER (SERVO MOTOR SPINDLE)	The parameter setting of an axis subject to spindle control with servo motor is illegal.
PW1111	ILLEGAL SPINDLE NUMBER (LIVE TOOL AXIS)	The spindle number (No. 11010) or spindle amplifier number (No. 3717) of an axis subject to spindle control with servo motor is illegally set.

Number	Message	Description
DS2003	ILLEGAL PARAMETER SETTING FOR LIVE TOOL AXIS (PMC AX-CTRL)	The axis subject to spindle control with servo motor was set as a PMC controlled axis.

Reference item

Manual name	Item name
CONNECTION MANUAL (FUNCTION) (This manual)	controlled axes
CONNECTION MANUAL (FUNCTION) (This manual)	2-path control
CONNECTION MANUAL (FUNCTION) (This manual)	Designation of spindle axes

10.19.2 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))

When multi-spindle control is enabled, select a spindle according to the specification of multi-spindle control along with the G96.1/G96.2/ G96.3 command. When a spindle is selected with address P, to specify address P, use parameter No. 3781 (P code for selecting a spindle). In any case, when a servo motor spindle has not been selected, alarm PS0602 is issued.
When multi-spindle control is disabled, it is possible to command G96.4 independently. When there is no servo motor spindle in the system, alarm PS0602 is issued.
In 2-path system, when it is specified from paths other than the path that a servo motor spindle belongs, alarm PS0602 is issued.
To turn the position control mode ON without performing spindle indexing, do so after canceling the SV speed control mode by issuing 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 having specified G96.2, check the completion of the movement by specifying G96.3 before specifying the next move command for the spindle. If the next move command is specified without confirming the completion of the movement, alarm PS0601 is issued. Similarly, if rigid tapping is specified without confirming the completion of the movement, alarm PS0445 is issued.
2 A block specifying G96.1, G96.2, or G96.3 must not contain any other command. If an axis command is specified in the same block, alarm PS0446 is issued.

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.
If SV speed control mode cancel, it is necessary to be commanded in the path to which the live tool axis belongs.

Example for specifying the start/cancellation of the SV speed control mode (when bit 3 (MPP) of parameter No. 3703 = 1)

Spindle name	Spindle selection P code (parameter No. 3781)	Servo motor spindle address
S1	P1	C

Specification by programming (SV speed control mode in-progress signal <Fn521>)

Programmed command	Start/cancellation of speed control mode	Operation
G96.4 P1 ;	Start (SV speed control mode in-progress signal (C) = 1)	The SV speed control mode is started (C).
M03 S100 P1 ;	:	Servo motor spindle C turns in the forward direction at 100 [min^{-1}].
:	:	:
G96.1 P1 R0 ;	Cancel (SV speed control mode in-progress signal (C) = 0)	The servo motor spindle C stops when C = 0 (spindle indexing).

Mode switching by signal

Programmed command	Start/cancellation of speed control mode	Operation
M15 ;	Start (SV speed control mode in-progress signal (C) = 1)	The SV speed control mode is started by the M code (C).
M03 S100 P1 ;	: (SV speed control mode in-progress signal (C) = 1)	Servo motor spindle C turns in the forward direction at 100 [min^{-1}].
: :	:	:
G96.1 P1 R0 ;	Cancel (SV speed control mode in-progress signal (C) = 0)	The servo motor spindle C stops when C = 0 (spindle indexing).

- Spindle indexing command during spindle rotation

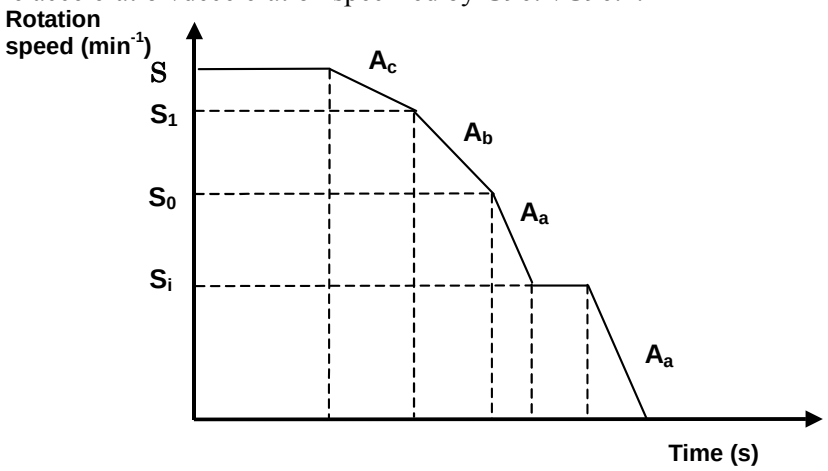
Issuing G96.1 or G96.2 with a position specified during spindle rotation causes the spindle to stop at the specified position.
Example)
M03 S1000 ;..... Rotation at S1000
G96.1 P1 R180. ;..... Stoppage of rotation at the 180° position

- 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 acceleration/deceleration

Shown below is the acceleration/deceleration specified by G96.1/G96.2.



S_1 : Parameter No. 11020 setting (acceleration/deceleration is switched at a rotation speed of S_1 (min^{-1}).)
S_0 : Parameter No. 11021 setting (acceleration/deceleration is switched at a rotation speed of S_0 (min^{-1}).)
S : Command-specified rotation speed (min^{-1})
Si : Spindle indexing speed (min^{-1}) parameter No. 11012
Aa : Parameter No. 11030 setting (acceleration/deceleration (min^{-1}/s) used between rotation speeds 0 and S_0 (span 1))
Ab : Parameter No. 11031 setting (acceleration/deceleration (min^{-1}/s) used between rotation speeds S_0 and S_1 (span 2))
Ac : Parameter No. 11032 setting (acceleration/deceleration (min^{-1}/s) used between S_1 and a specified speed S (span 3)))

- Example program commands

(1) A move command is issued to the spindle, using G96.2. The spindle starts moving, and the execution of the next block begins.
The spindle keeps moving even when any other block is being executed.
(The spindle indexing signal SPP<Fn522> is "1" during spindle indexing.)
(2) When another command is issued to the spindle, G96.3 is used to heck in advance whether the spindle has finished moving.
If the spindle is still moving (the spindle indexing signal is on), the CNC enters a wait state. If the spindle has finished moving, the command is issued to cause the spindle to start moving.

Example: Command not waiting for spindle indexing to finish and command checking whether spindle indexing has finished (parameter No. 3781 (S1) = "1")

Program command	Operation
G96.2 P1 R270.0 ;	Command not waiting for spindle indexing to finish. The first spindle S1 moves to 270.0.
G01 X10.0 Y20.0 F1000. ;	Starts cutting feed. No wait for spindle indexing to finish.
G02 X50.0 Y100.0 R50.0 ;	Starts circular interpolation. No wait for spindle indexing to finish.
G96.3 P1 ;	Checks whether spindle indexing has finished.
M29 S100 P1 ;	Starts rigid tapping if the spindle indexing signal is "0".
G84 X10.0 Y 20.0 R-5.0 Z-20.0 ;	Waits for rigid tapping command if the spindle indexing signal is "1".

Indexing by machine coordinates

In the function for spindle control with servo motor, the following operations can be selected:

(1) When the speed control mode is off, indexing to machine coordinate 0.000 is performed.
(2) When G code-based spindle indexing has been performed, an R command indicates a machine coordinate value.

Spindle indexing operation

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 signal <Gn521> OFF (1 t 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.

Notes

(1) If G96.2 (not waiting for spindle indexing to finish) is issued, G96.3 must be issued to check whether spindle indexing has finished. Be sure to issue G96.3 after G96.2. If a movement along an axis is specified without checking (without issuing the G96.3 command), alarm PS0601 is issued. Similarly, if rigid tapping is specified, alarm PS0445 is issued.
(2) If it has not been checked, for example, because of a reset, whether spindle indexing has finished after G96.2 (not waiting for spindle indexing to finish) is issued, the SV speed control mode is not canceled.
(3) If the spindle stops rotating because of spindle indexing, the spindle speed command output becomes 0. To cause the spindle to restart rotating, place the spindle in the SV speed control mode, and then, issue an S command.
(4) Spindle indexing is enabled only in the SV speed control mode.
(5) When the spindle indexing speed (parameter No. 11012) is 0, the acceleration/deceleration switching speed (1st step) (parameter No. 11020) is the spindle indexing speed. When the acceleration/deceleration switching speed is also 0, the maximum speed (parameter No. 3741) of each spindle that corresponds to gear 1 is the spindle indexing speed.
(6) The G96.* P command is invalid for settings other than multi-spindle control type P.
(7) The SV speed control mode during automatic operation must be switched by a command from the path to which it belongs. When it is specified from the other path, alarm PS0602 is issued.

Signal

Spindle indexing signal for each axis SPP1 to SPP5 <Fn522.0 to Fn522.4>

[Classification] Output signal

[Function] These signals inform that spindle indexing for the respective axes is in progress.

[Output cond.] The signals are "1" if:
- Spindle indexing for the respective axes has not finished.

The signals are "0" if:
- Spindle indexing for the respective axes has finished.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Fn522				SPP5	SPP4	SPP3	SPP2	SPP1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3703					MPP			

[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.

#3 MPP In multi-spindle control, a spindle selection using a programmed command instead of using the signals (SWS1 to SWS2<G027.0, 1>) is:
0: Not made.
1: Made.

NOTE
When this parameter is set to 1, set parameter No. 3781 at the same time.

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.

NOTE
1 This parameter is valid if bit 3 (MPP) of parameter No. 3703 is set to 1.
2 If this parameter is set to 0, the corresponding spindle cannot be selected by a P code.
3 Under 2-path control, the P code specified here is valid for each path.
For instance, if the P code to select the first spindle of path 2 is set to 21, specifying S1000 P21; in path 1 causes the first spindle of path 2 to be rotated at S1000.

NOTE
4 Identical P code values cannot be used for different spindles. (Identical P code values cannot be used even if the paths are different.)
5 When this parameter is used (when bit 3 (MPP) of parameter No. 3703 is set to 1), the spindle command selection signal is invalid.
6 To use this parameter, enable multi-spindle control (bit 3 (MSP) of parameter No. 8133 is 1).

	#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.

11012	Spindle indexing speed for each axis

[Input type] Parameter input
[Data type] Word axis
[Unit of data] min^{-1}
In spindle control with servo motor, set the spindle indexing speed for each axis.
If 0 is set, the spindle indexing speed is assumed to be the setting of parameter No. 11020 (speed (S_0) for switching acceleration/ deceleration for each axis).

11020	Acceleration/deceleration switching speed (1st) for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] min^{-1}
[Valid data range] 0 to 99999999
This parameter sets the speed at which acceleration/deceleration is switched during rotation control in spindle control with servo motor. (First step)

Alarm and message

Number	Message	Description
PS0445	ILLEGAL AXIS OPERATION	A positioning command is specified in the speed control mode. Check the SV speed control mode in-progress signal <Fn521>.
PS0446	ILLEGAL COMMAND IN G96.1/G96.2/G96.3/G96.4	G96.1, G96.2, G96.3, or G96.4 is specified with another command in the same block. Correct the program.
PS0601	ILLEGAL AXIS OPERATION FOR LIVE TOOL AXIS	A movement command was specified for an axis subject to spindle control with servo motor. Correct the program.
PS0602	ILLEGAL AXIS OPERATION FOR LIVE TOOL AXIS	An axis subject to spindle control with servo motor is not correctly selected.

10.19.3 Rigid Tapping with Servo Motor

Format

The command format for this type of rigid tapping is the same as for the conventional type of rigid tapping.
For details, refer to the chapter of "RIGID TAPPING" in the Operator's Manual (Lathe System) (B-64304EN-1) or Operator's Manual (Machining Center System) (B-64304EN-2).

NOTE
Before rigid tapping can be specified, the SV speed control mode for the servo motor spindle must be canceled. If rotation is in progress, use G96.1/G96.2 to cancel the SV speed control mode. The mode of the servo motor spindle can be checked by checking the SV speed control mode in-progress signal (SVREV<Fn521>).
If rigid tapping is specified in the SV speed control mode, alarm PS0445 is issued.

Rigid tapping specification

- Feedrate

For rigid tapping, the feedrate of a drilling axis is one specified in an F command, and that of the spindle is S × the amount of movement per live tool axis (servo motor spindle) rotation [deg/min] (parameter No. 11011).

Feed per minute and feed per revolution are detailed later.
During rigid tapping, the spindle speed is limited by a parameter for specifying the maximum cutting feedrate for the axis used as a live tool axis, that is, parameter No. 1430 (or No. 1432 if acceleration/ deceleration before interpolation is enabled).
Normally, the parameter of the maximum cutting feedrate (parameter No. 1430) (parameter No. 1432 when acceleration/deceleration before interpolation is enabled) can be set to up to 999999.999 [deg/min] (equivalent to S2778 [min^{-1}]). However, for an axis set as the live tool axis (bit 3 (IRC) of parameter No. 1408 = 1) that is used for rigid tapping, the maximum feedrate is limited to ten times as large as the setting of the maximum cutting feedrate parameter.
Example:
Maximum cutting feedrate parameter No. 1430 = 360000
Limit to the maximum spindle speed 360000×10 = 3600000 [deg/min] (S10000 [min^{-1}])

⚠ CAUTION
Make the tapper thread pitch equal to one specified by the program (F, S). Otherwise, the tool or workpiece may be damaged.

Acceleration/deceleration control

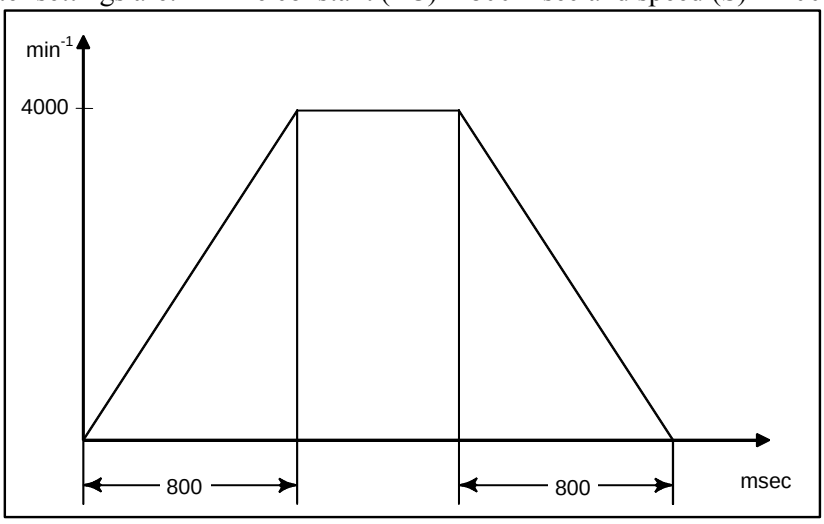
- Acceleration/deceleration after interpolation

Unlike conventional rigid tapping (with a spindle motor), rigid tapping with a servo motor allows the application of linear acceleration/deceleration with constant acceleration/deceleration time or bell-shaped acceleration/deceleration.
Resetting bit 0 (SRBx) of parameter No. 11001 to "0" makes it possible to apply linear acceleration/deceleration after interpolation of constant acceleration time type. Setting the bit to "1" makes it possible to apply bell-shape acceleration/deceleration after interpolation of constant acceleration time type.
The time constant for each gear is specified in parameter Nos. 11060 to 11063. If bit 2 (TDR) of parameter No. 5201 = "1", the tool extraction time constant for each gear is specified in parameter

Nos. 11065 to 11068. Specify each of these parameters for the live tool axis (servo motor spindle) used in rigid tapping.
The acceleration/deceleration types and time constants used for drilling axes are set to the same values as for live tool axes (servo motor spindles).

NOTE
This type of rigid tapping and conventional rigid tapping (with a spindle motor) differ in time constant setting.

Example: Parameter settings are: Time constant (TC) = 800 msec and speed (S) = 4000 min^{-1}



- Acceleration/deceleration before interpolation

In this type of rigid tapping, when advanced preview control can be used, rigid tapping can be specified in the mode for acceleration/deceleration before look ahead interpolation to allow acceleration/deceleration before look ahead interpolation during rigid tapping. Acceleration/deceleration before look ahead interpolation is enabled when the advanced preview control mode is set to on. For the advanced preview control function, see "High-Speed High-Precision Function (Advanced Preview Control)".
Parameter No. 11050 is used to specify the maximum allowable acceleration value for acceleration/deceleration interpolation used in rigid tapping. Parameter No. 11051 is used to specify the acceleration change time for bell-shape acceleration/deceleration before interpolation. As for acceleration/deceleration before interpolation used in rigid tapping, the maximum allowable acceleration value is 100000 [deg/s^2].
It is possible to change a speed of up to S1000 [min^{-1}] (equivalent to 360000 [deg/min] in 60 [ms].
The time constant (parameter No. 11052) for cutting feed acceleration/deceleration after interpolation usable in the "acceleration/deceleration before look-ahead interpolation" mode is a constant-time type.

NOTE
Specify the same time constant for both drilling and live tool axes (servo motor spindle). Otherwise, it is likely that the machine may malfunction.

Spindle indexing

This function does not allow the spindle orientation function to be performed at the start of rigid tapping. Before specifying rigid tapping, perform spindle indexing to the position at which tapping is to be performed. For details, see "Spindle Indexing Function" described previously.

Notes

If the pitch is very small or the amount of travel along the drilling axis is large, the amount of travel along the rotation axis becomes large, possibly resulting in alarm PS0003.

Parameter

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.

	#7	#6	#5	#4	#3	#2	#1	#0
5201						TDR		

[Input type] Parameter input
[Data type] Bit path

#2 TDR In rigid tapping, the cutting time constant is:
0: Set in the same parameter for both cutting and extraction.
(Parameter Nos. 11060 to 11063) * For rigid tapping using a servo motor spindle
1: Set in different parameters separately for cutting and extraction.
(Parameter Nos.11060 to 11063: Time constant for cutting)
(Parameter Nos. 11065 to 11068: Time constant for extraction)
* For rigid tapping using a servo motor spindle

NOTE
In case of rigid tapping using a spindle, time constants are set in different parameters.

	#7	#6	#5	#4	#3	#2	#1	#0
11001								SRBx

[Input type] Parameter input
[Data type] Bit axis

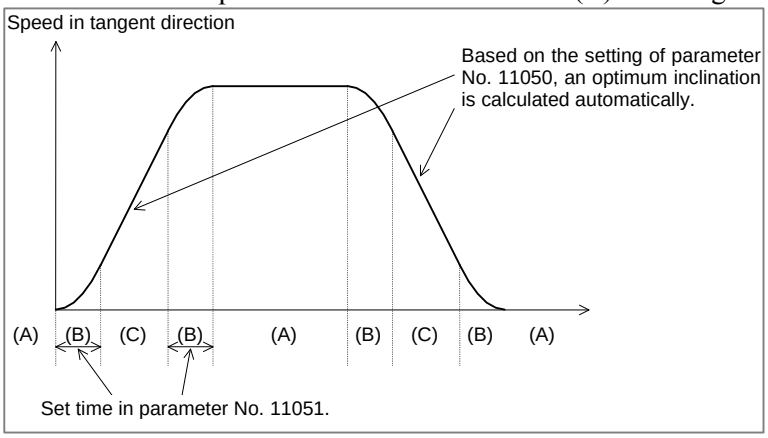
#0 SRBx Acceleration/deceleration after interpolation in cutting feed during rigid tapping with servo motor is:
0: Linear acceleration/deceleration.
1: Bell-shaped acceleration/deceleration.

11050	Maximum allowable acceleration rate in acceleration/deceleration before interpolation for each axis in rigid tapping

[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, 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.

11051	Acceleration change time of bell-shaped acceleration/deceleration before interpolation in rigid tapping

[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 federate (A) to the state of constant acceleration/deceleration (C) at the acceleration rate calculated from the acceleration rate set in parameter No. 11050: time of (B) in the figure below).



11052	Time constant for acceleration/deceleration after cutting feed interpolation in the acceleration/deceleration before interpolation mode in rigid tapping

[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 advanced preview 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.

11060	Time constant for acceleration/deceleration after cutting feed interpolation in rigid tapping (first gear)
11061	Time constant for acceleration/deceleration after cutting feed interpolation in rigid tapping (second gear)
11062	Time constant for acceleration/deceleration after cutting feed interpolation in rigid tapping (third gear)
11063	Time constant for acceleration/deceleration after cutting feed interpolation in rigid tapping (fourth gear)

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
Set the time constant of rigid tapping with servo motor in these parameters (Nos. 11060 to 11063).
Set these parameters with a live tool axis in rigid tapping.

11065	Time constant for acceleration/deceleration deceleration after cutting feed interpolation in rigid tapping extraction (first gear)
11066	Time constant for acceleration/deceleration deceleration after cutting feed interpolation in rigid tapping extraction (second gear)
11067	Time constant for acceleration/deceleration deceleration after cutting feed interpolation in rigid tapping extraction (third gear)
11068	Time constant for acceleration/deceleration deceleration after cutting feed interpolation in rigid tapping extraction (fourth gear)

[Input type] Parameter input
[Data type] Word axis
[Unit of data] msec
[Valid data range] 0 to 4000
When bit 2 (TDR) of parameter No. 5201 is set to 1, time constants for extraction of rigid tapping with servo motor can be set in these parameters (Nos. 11065 to 11068).
Set these parameters with a live tool axis in rigid tapping.

Alarm and message

Number	Message	Description
PS0003	TOO MANY DIGIT	The permissible number of word digits in NC commands is exceeded. The permissible number of digits varies depending on the function and address.
PS0445	ILLEGAL AXIS OPERATION	A positioning command is specified in the speed control mode. Check the SV speed control mode in-progress signal (Fn521).

10.19.4 Feed per Revolution

Overview

The function for spindle control with servo motor allows a feed per revolution to be specified.
From the separate detector connected to the spindle, the rate of feed per revolution is obtained. When the detector built into the servo motor is to be used, the feedrate is obtained based on the servo motor speed and gear ratio.
Which detector to use is specified by bit 1 (OPTx) of parameter No. 1815.

NOTE
If the machine is equipped with a live tool axis and spindle with servo motor, using rotation commands requires the multi-spindle control function.
As for gear change, use T type gear change.
See Section, "MULTI-SPINDLE CONTROL," in this manual for explanations about multi-spindle control.

Format

- Feed per revolution, threading, and constant surface speed control

The command format for feed per revolution explained above is the same as for the ordinary types of feed per revolution.

Examples

When the second spindle (C-axis) is used for rotation control and position control with the second spindle assumed to be a servo motor spindle
(The G code system A in lathe system is selected.)

- Feed per revolution

Program command	Operation
M*** ;	The C axis rotation control mode is turned ON.
M03 S100 P2 ;	The servo motor spindle (C axis) rotates at 100 min-1.
G99 G01 Z-100. F10. ;	The Z axis moves at a feed-per-revolution speed of 1000 mm/min.
:	:
M*** ;	The C axis position control mode is set to ON (rotation control mode OFF). The spindle stops with C = 0.000.

Parameters

	#7	#6	#5	#4	#3	#2	#1	#0
1815							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 pulse coder is not used.
1: A separate pulse coder is used.

2455	Integral part (α) of the number of pulses for one rotation

[Data type] Word axis
[Valid data range] 0 to 32767
If 0 is specified, no threading counter is output.
If the specified range is exceeded, an invalid parameter alarm SV0417 is issued.
(Detail number of diagnosis display No. 352: 4553)

2456	Exponential part (β) of the number of pulses for one rotation

[Data type] Word axis
[Valid data range] 0 to 12
If the specified range is exceeded, an invalid parameter alarm SV0417 is issued.
(Detail number of diagnosis display No. 352: 4563)

Number of pulses per detector rotation (Np) = $\alpha \times 2^{\beta}$
Determine α and β so that the above expression is satisfied.
For a closed system is in use, specify a value that corresponds to separate detector pulses.

10.19.5 Spindle Output Control with PMC

Overview

The “spindle control with servo motor” function enables the PMC to be sued for spindle output control.

How to specify

After the start of the SV speed control mode, this function can be specified in the same way as in ordinary spindle control (with a spindle motor).
Refer to Section, “SPINDLE OUTPUT CONTROL BY THE PMC,” in this manual for detailed descriptions.

NOTE
For the “spindle control with servo motor” function, the maximum motor speed is one specified in parameter No.11015.

NOTE
Volume 1 : Page 1 to 1008 / Volume 2 : P. 1009 to 1648

INDEX

<Number>

2ND REFERENCE POSITION RETURN / 3RD, 4TH REFERENCE POSITION RETURN 332
2-PATH CONTROL 578
2-path Functions.....583
8-Level Data Protection Function 1178

<A>

Absolute Position Detection.....54
Acceleration/Deceleration after Interpolation 849
ACCELERATION/DECELERATION CONTROL 555
Acceleration-Based Speed Control in Circular Interpolation.....521
ACTUAL SPINDLE SPEED OUTPUT (T SERIES) ..763
Adding workpiece coordinate systems (G54.1 or G54) (M series).....98
ADVANCED PREVIEW CONTROL (T SERIES) / AI ADVANCED PREVIEW CONTROL (M SERIES) / AI CONTOUR CONTROL (M SERIES) 1587
ALARM SIGNALS.....209
ARBITRARY ANGULAR AXIS CONTROL....144,1571
Automatic Acceleration/Deceleration 555
Automatic Alteration of Tool Position Compensation (T Function).....1017
Automatic coordinate system setting 99
Automatic Corner Override (M Series).....511
AUTOMATIC OPERATION 381
AUTOMATIC REFERENCE POSITION RETURN AND RETURN FROM THE REFERENCE POSITION 327
Automatic Setting for Grid Position Matching.....118
Automatic Slave Axis Parameter Setting 121
Automatic Tool Length Measurement (M Series).....1630
AUTOMATIC TOOL LENGTH MEASUREMENT (M SERIES) / AUTOMATIC TOOL OFFSET (T SERIES) 1260
AUTOMATIC TOOL LENGTH MEASUREMENT (M SERIES)/AUTOMATIC TOOL OFFSET (T SERIES) 1630
Automatic Tool Offset (T Series).....1631
AUXILIARY FUNCTION.....682
AUXILIARY FUNCTION LOCK.....694
AUXILIARY FUNCTION/2ND AUXILIARY FUNCTION 682,1598
Available diagnosis number.....1448
Axis Configuration for Axis Synchronous Control.....113
AXIS CONTROL.....1
AXIS SYNCHRONOUS CONTROL 113,1565
Axis Synchronous Control Torque Difference Alarm..120

<B>

Backlash Compensation 28
BALANCE CUTTING 602
Bell-Shaped Acceleration/Deceleration after Cutting Feed Interpolation (M Series) 565
Bi-directional Pitch Error Compensation 40

<C>

CANNED CYCLE (T SERIES) / MULTIPLE REPETITIVE CANNED CYCLE (T SERIES).....1111
CANNED CYCLE (T SERIES)/MULTIPLE REPETITIVE CANNED CYCLE (T SERIES).....1619
CANNED CYCLE FOR DRILLING.....1096,1617
CANNED GRINDING CYCLE.....1620
CANNED GRINDING CYCLE (FOR GRINDING MACHINE).....1121
Cautions on 2-path Control 585
CHAMFERING AND CORNER ROUNDING (T SERIES) 1148,1626
Checking input data.....1456
Checking the Stored Stroke Limit during the Time from Power–on to the Reference Position Establishment 194
Chuck and Tail Stock Barrier (T Series) 197
CHUCK/TAIL STOCK BARRIER (T SERIES) 1574
CIRCULAR INTERPOLATION 432,1583
CNC Data Display, Setup, and Input/Output 583
CNC READY SIGNALS 179
CNC Screen Dual Display.....1203
Command Format.....870
Commands for Feed Per Minute and Feed Per Revolution 849
Compact-Type MDI Key Input Function 1220
COMPENSATION VALUE INPUT.....1292
Composite Control 613
Connection Among Spindle, Spindle Motor, and Position Coder.....844
CONSTANT SURFACE SPEED CONTROL 755,1600
Continuous Threading (T Series) 447
Controlled Axes Detach 10
CONTROLLED AXIS 1
COORDINATE SYSTEM ROTATION 1145
Corner Control 567
CORRESPONDENCE OF ROTARY SCALE WITHOUT ROTARY DATA 368
Cs CONTOUR CONTROL.....797,1602
Cs Contour Control Axis Coordinate Establishment....818
Cs Contour Control Torque Limit Skip.....815
CUSTOM MACRO.....1075,1614
Cutter Compensation (M Series) and Tool Nose Radius Compensation (T Series).....1027
CUTTER COMPENSATION/TOOL NOSE RADIUS COMPENSATION.....1608
Cutting Feedrate Clamp 499
CYCLE START/FEED HOLD 381
CYLINDRICAL INTERPOLATION 452

NOTE
Volume 1 : Page 1 to 1008 / Volume 2 : P. 1009 to 1648

<D>

DATA SERVER FUNCTION 1646
DECIMAL POINT PROGRAMMING/POCKET CALCULATOR TYPE DECIMAL POINT PROGRAMMING 1059
DEFINITION OF WARNING, CAUTION, AND NOTE s-1
DESIGNATION OF SPINDLE AXES 701
Diagnosis Display 867
DIAGNOSIS FUNCTION 1429
Differences between Pitch Error Compensation, Simple Straightness Compensation, and Gradient Compensation (Reference).....47
DIFFERENCES FROM Series 0*i*-C 1560
DIRECT DRAWING DIMENSIONS PROGRAMMING (T SERIES) 1150,1627
DIRECT OPERATION BY C LANGUAGE EXECUTOR 425
DISPLAY/SET.....1166
DISPLAY/SET/EDIT.....1166
DISTANCE CODED LINEAR SCALE INTERFACE339
DNC OPERATION.....419
Dry Run.....389

<E>

Each axis workpiece coordinate system preset signals.104
EDIT 1217
ELECTRONIC GEAR BOX (M SERIES) 156
EMBEDDED ETHERNET FUNCTION 1407
EMBEDDED ETHERNET PORT AND PCMCIA ETHERNET CARD 1407
EMERGENCY STOP 177
Environment for making trouble diagnosis message..1449
ERROR COMPENSATION 22
EXACT STOP / EXACT STOP MODE / TAPPING MODE / CUTTING MODE.....413
Example of setting the FOCAS2/Ethernet function ...1411
Example of setting the FTP file transfer function 1416
Explanation 1228,1235
EXTENDED EXTERNAL MACHINE ZERO POINT SHIFT 1390
EXTENDED FUNCTION OF THE DISTANCE CODED LINEAR SCALE INTERFACE.....361
EXTERNAL DATA INPUT 1377,1642
External Deceleration.....516
EXTERNAL I/O DEVICE CONTROL 1254
EXTERNAL KEY INPUT.....1395
EXTERNAL SUBPROGRAM CALL (M198) 1648
External Touch Panel Interface.....1188
EXTERNAL WORKPIECE NUMBER SEARCH 1393

<F>

Feed Forward in Rapid Traverse.....572
Feed Per Minute.....499
Feed per Revolution.....1007
Feed Per Revolution/Manual Feed Per Revolution 501
FEEDRATE CONTROL.....496
FEEDRATE CONTROL/ACCELERATION AND DECELERATION CONTROL.....496
Feedrate override.....509
Follow-up 16
FS10/11 Format Command 854
FSSB Setting 60

<G>

G Code List in the Lathe System.....1061
G Code List in the Machining Center System 1063
G CODE SYSTEM 1061
GENERAL PURPOSE RETRACT.....491
Gradient Compensation.....37
Guidance table for machine alarm diagnosis.....1450

<H>

HELICAL INTERPOLATION 448,1584
High Precision and High Speed Functions (Advanced Preview Control (T Series)/AI Advanced Preview Control (M Series)/AI Contour Control (M Series)/AI Contour Control II (M Series)).....522
HIGH-SPEED M/S/T/B INTERFACE.....697
High-speed Skip Signal 1274
Hypothetical Cs Axis Control 646

<I>

I/O Link β MANUAL HANDLE INTERFACE 276
In the Case of a Rotary Axis A Type 377
In the Case of a Rotary Axis B Type whose Movable Range is over One Rotation.....375
In the Case of a Rotary Axis B Type whose Movable Range is under One Rotation.....369
INCH/METRIC CONVERSION 1070
Increment System.....4
INDEX TABLE INDEXING (M SERIES).....1128
IN-FEED CONTROL (FOR GRINDING MACHINE) (M SERIES) 1120
Inner corner automatic override (G62).....512
In-position check.....568,1645
In-position check independently of feed/rapid traverse570
In-position check signal 567
Input of tool offset value measured (T Series) 1292
Input of Tool Offset Value Measured B (T Series) 1293,1635
INPUT/OUTPUT OF DATA 1243
Inputting data in the sheet for multi-languages 1461
Inputting guidance data 1454
Install.....1450
INTERFACE BETWEEN CNC AND PMC 1465
Internal circular cutting feedrate change 514
INTERPOLATION FUNCTION 426
Interruption Type Custom Macro.....1093,1616
Inverse Time Feed (M Series).....505

NOTE
Volume 1 : Page 1 to 1008 / Volume 2 : P. 1009 to 1648

<J>

JERK CONTROL (M Series).....572
JOG FEED/INCREMENTAL FEED 256
Jump from CNC guidance table to MTB's guidance table 1458

<K>

Kind of additional alarm and operator message 1448

<L>

Limitation.....1234
Linear Acceleration/Deceleration after Cutting Feed Interpolation.....562
LINEAR INTERPOLATION.....430
LINEAR INTERPOLATION (G28, G30, G53).....483
LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL) 351
LIST OF ADDRESSES 1465
LIST OF SIGNALS 1502
List of Signals (In Order of Addresses).....1541
List of Signals (In Order of Functions) 1502
List of Signals (In Order of Symbols) 1523
Local Coordinate System 109,1564
LOG SCREEN OF THE EMBEDDED ETHERNET FUNCTION 1426
Look-Ahead Smooth Bell-Shaped Acceleration/Deceleration before Interpolation 576

<M>

M29 and G84/G74 are specified in the same block.....887
M29 and G84/G88 are specified in the same block.....897
MACHINE ALARM DIAGNOSIS.....1448
Machine Coordinate System 92
Machine Lock 388
MACHINE OPERATION MENU 1228
MACHINE OPERATION MENU TOOL.....1235
MACHINING CONDITION SELECTION FUNCTION 235,1574
MACHINING QUALITY LEVEL ADJUSTMENT (M Series) 241
MACRO COMPILER/MACRO EXECUTER.....1146
MAINTENANCE SCREEN FOR EMBEDDED ETHERNET FUNCTION 1422
Making a file to input trouble diagnosis messages.....1451
Making a memory card format file.....1457
Making messages for multi-languages 1459
Making sheets for multi-languages 1459
Making trouble diagnosis messages 1453
MALFUNCTION PREVENT FUNCTIONS.....243
MANUAL ABSOLUTE ON AND OFF 1581
MANUAL ABSOLUTE ON/OFF.....393
MANUAL HANDLE FEED 262,1576
MANUAL HANDLE INTERRUPT 271
MANUAL HANDLE RETRACE.....279
MANUAL INTERVENTION AND RETURN.....422
MANUAL OPERATION.....256
MANUAL REFERENCE POSITION RETURN 299,1577
MDI Key Setting 1219
MEASUREMENT 1259
MEMORY COMMON TO PATHS.....675
Memory Protection Keys 1217
Memory Protection Signal For CNC Parameter 1218,1629
Method of Using Heidenhain Rotary Scale RCN223, 723 and 220.....378
Methods of Alarm Recovery by Synchronization Error Check 119
Mirror Image 14
MIRROR IMAGE FOR DOUBLE TURRET (T SERIES) 1126
Miscellaneous.....1616
MODE SELECTION 216
Multi Spindle Control.....856
MULTIPLE M COMMANDS IN A SINGLE BLOCK695
MULTIPLE RESPECTIVE CANNED CYCLE FOR TURNING (T SERIES).....1621
MULTI-SPINDLE CONTROL.....826,1603
Multi-step Skip.....1279

<N>

Name of Axes.....2
NANO SMOOTHING (M SERIES) 484
NORMAL DIRECTION CONTROL (M SERIES) 480
Notes 866,923
Notes on interface with the PMC 880
Notice.....1461

<O>

Offset.....1011
ONE TOUCH MACRO CALL 1400
One-digit F Code Feed (M Series) 502
Operation on the FOCAS2/Ethernet setting screen 1408
Operation on the FTP file transfer setting screen 1412
OPERATOR ERROR PREVENT FUNCTIONS.....246
Optimum Acceleration/Deceleration for Rigid Tapping860
OPTIONAL ANGLE CHAMFERING AND CORNER ROUNDING (M SERIES) 1147,1625
OPTIONAL BLOCK SKIP/ADDITION OF OPTIONAL BLOCK SKIP.....395
Outline.....1433,1448
Outputting the Movement State of an Axis 13
Override 505,851
Override cancel 511
OVERTRAVEL CHECK.....180
Overtravel Signals.....180
Overview.....1228,1235

<P>

Parameter 902,1233,1446
Parameter Check Sum Function 1191
Parameters Related to Servo.....48
PART PROGRAM STORAGE SIZE / NUMBER OF REGISTERABLE PROGRAMS.....1069

NOTE
Volume 1 : Page 1 to 1008 / Volume 2 : P. 1009 to 1648

PATH INTERFERENCE CHECK....593
PATH INTERFERENCE CHECK (T SERIES (2-PATH CONTROL))....1591
PATH SELECTION/DISPLAY OF OPTIONAL PATH NAMES....679
PATH SINGLE BLOCK CHECK FUNCTION....678
PATH SPINDLE CONTROL....665
PATTERN DATA INPUT....1152
PMC AXIS CONTROL....1309,1636
PMC CONTROL FUNCTION....1309
POLAR COORDINATE INTERPOLATION (T SERIES)....449,1585
Polygon Turning....455
POLYGON TURNING (T SERIES)....454
Polygon Turning with Two Spindles....462
Position Control Loop Gain Parameter Switching....874
Position Switch....19
POSITIONING....426
POWER MATE CNC MANAGER....1647
PREFACE....p-1
PREPARATIONS FOR OPERATION....177
PROGRAM COMMAND....1059
PROGRAM CONFIGURATION....1067
PROGRAM RESTART....397
PROGRAMMABLE PARAMETER INPUT (G10)....1647

<R>

Rapid Traverse Bell-shaped Acceleration/Deceleration560
Rapid traverse block overlap....559
Rapid traverse override....505
Rapid Traverse Rate....497
READER/PUNCHER INTERFACE....1243
Reference Position Established by the G00 Command 361
REFERENCE POSITION ESTABLISHMENT....299
Reference Position Establishment by Jog Feed....365
Reference Position Return....853
REFERENCE POSITION SETTING WITH MECHANICAL STOPPER....334
REFERENCE POSITION SETTING WITHOUT DOG....320
Related NC parameters....1414,1420
RESET AND REWIND....384,1580
RESTART OF THE EMBEDDED ETHERNET....1421
Restrictions....1447
RETRACE (M SERIES)....403
RETRACTION FOR RIGID TAPPING....414
RIGID TAPPING....843
Rigid Tapping Specification....848
Rigid Tapping with Servo Motor....1002
Rigid Tapping with Spindle of Another Path (T Series (2-ptah Control))....856
Rotary Axis Roll Over....111
Run Hour and Parts Count Display....1166,1627

<S>

SCALING (M SERIES)....1138
Screen erasure function and automatic screen erasure function....1212,1628
Screen Hard Copy Function....1214
Screen Switching at Path Switching....1211
Screen Switching by Mode....1208
SEQUENCE NUMBER COMPARISON AND STOP 397
SEQUENCE NUMBER SEARCH....1645
SERIAL/ANALOG SPINDLE CONTROL....719,1599
Series 0*i*-C-compatible setting....76
Series 0*i*-D-dedicated setting....60
Servo off/Mechanical Handle Feed....17
SERVO WARNING INTERFACE....1429
SETTING EACH AXIS....2
Setting Method by Rotary Axis Type and Movable Range....368
Setting of the FOCAS2/Ethernet Function....1408
Setting of the FTP File Transfer Function....1411
SETTING UNIT....1561
Setting up DHCP....1417
Setting up DNS....1416
Setting Up the DNS/DHCP Function....1416
SETTING UP THE EMBEDDED ETHERNET FUNCTION....1408
SETTINGS RELATED TO SERVO-CONTROLLED AXES....48
SETTINGS RELATED WITH COORDINATE SYSTEMS....92
Signal....875,1234,1447
Signals for the rigid tapping function....875
Signals related to gear switching....876
Signals related to S code output....875
Signals related to the addition of multi spindle control 877
SIMPLE SPINDLE SYNCHRONOUS CONTROL (M SERIES)....949
Simple Straightness Compensation (M Series)....33
Simultaneous Two Path Program Editing....1224
Single Block....391
SINGLE DIRECTION POSITIONING (M SERIES)....427,1581
SKIP FUNCTION....1267,1633
Smooth Backlash....30
Software Operator's Panel....1170
Specifying G84/G74 for rigid tapping by parameters ..891
Specifying G84/G88 for rigid tapping by parameters ..899
Specifying the Rotation Axis....7
Speed Command Extension in Least Input Increment C....552
Speed Control with Change of Acceleration on Each Axis....572
Speed Display Function of a Milling Tool with Servo Motor....1205
SPINDLE ANALOG OUTPUT....715
SPINDLE CONTROL WITH SERVO MOTOR ..970,972
Spindle Indexing Function....996
SPINDLE ORIENTATION....945
SPINDLE OUTPUT CONTROL BY THE PMC....749

NOTE
Volume 1 : Page 1 to 1008 / Volume 2 : P. 1009 to 1648

Spindle Output Control with PMC 1008
SPINDLE OUTPUT SWITCHING 948
SPINDLE POSITIONING (T SERIES) 764,1601
SPINDLE SERIAL OUTPUT 705
SPINDLE SPEED CONTROL 722
SPINDLE SPEED FLUCTUATION DETECTION (T SERIES) 960
SPINDLE SPEED FUNCTION 700
SPINDLE SPEED FUNCTION (S CODE OUTPUT) 700
SPINDLE SYNCHRONOUS CONTROL 927
SPINDLE WARNING INTERFACE 1431
START LOCK / INTERLOCK 210
STATUS OUTPUT SIGNAL 222
Stored Pitch Error Compensation 22,1562
STORED STROKE CHECK 1572
Stored Stroke Check 1 182
Stored Stroke Check 2, 3 187
Stroke Limit Check Before Move 206
Stroke Limit External Setting (M Series) 196
Structure of the file to input trouble diagnosis messages 1452
SUPERIMPOSED CONTROL 650
SUPERIMPOSED CONTROL (T SERIES (2-PATH CONTROL)) 1596
SWITCHING BETWEEN THE EMBEDDED ETHERNET DEVICES 1421
Synchronization Error Check 118
Synchronization Establishment 116
Synchronous Control 607
SYNCHRONOUS CONTROL AND COMPOSITE CONTROL 605
SYNCHRONOUS CONTROL AND COMPOSITE CONTROL (T SERIES (2-PATH CONTROL)) 1592
SYNCHRONOUS, COMPOSITE, AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND 663

<T>

TANDEM CONTROL 136
Temporary Absolute Coordinate Setting 89
TESTING A PROGRAM 388
THREADING 435
THREADING CYCLE RETRACT (CANNED CUTTING CYCLE/MULTIPLE REPETITIVE CANNED CUTTING CYCLE) (T SERIES) 1583
Threading Cycle Retract (Canned Cycle) (T Series) 440
Threading Cycle Retract (Multiple Repetitive Canned Cycle) (T Series) 443
Timing Charts for Rigid Tapping Specification 882
Timing of the M code for unclamping (T series) 901
Timing to cancel rigid tapping mode 901
TOOL COMPENSATION 1027
Tool Compensation Memory 1022,1606
TOOL FUNCTIONS 1009,1604
TOOL FUNCTIONS OF M SERIES 1021
TOOL FUNCTIONS OF T SERIES 1009
Tool Geometry Offset and Tool Wear Offset 1010
Tool Length Compensation 1035
Tool Length Compensation Shift Types 1038
TOOL LENGTH MEASUREMENT (M SERIES) 1259
TOOL LIFE MANAGEMENT 1042
Tool Offset 1010
Torque Limit Skip Function 1286
Touch Panel Check Signal 1201
Touch Panel Control 1183
Translating data used with the former series(Series 0*i* /0*i* Mate-B/C, Series 16*i* /18*i* /21*i*-B) 1461
TROUBLE DIAGNOSIS 1433
Trouble diagnosis graphic screen 1443
Trouble diagnosis guidance screen 1435
Trouble diagnosis monitor screen 1437
Trouble diagnosis parameter screen 1441
Trouble forecast level setting screen (only for servo axis) 1444
Two Path Display 1222
TWO PATH DISPLAY AND EDIT 1222

<U>

UNEXPECTED DISTURBANCE TORQUE DETECTION FUNCTION 225
Uninstall 1450

<V>

Variable Lead Threading (T Series) 447
VRDY OFF ALARM IGNORE SIGNAL 223

<W>

WAITING M CODES 591
WAITING M CODES (T SERIES (2-PATH CONTROL)) 1590
When M29 is specified before G84/G74 883
When M29 is specified before G84/G88 895
Workpiece coordinate system 94,1563
Workpiece coordinate system preset 97
Workpiece coordinate system shift (T series) 100
Workpiece Coordinate System/Addition of Workpiece Coordinate System Pair 94
Workpiece Origin Offset Measurement Value Direct Input 1306

<Y>

Y AXIS OFFSET (T SERIES) 1607

Revision Record

FANUC Series 0*i*-MODEL D/Series 0*i* Mate-MODEL D CONNECTION MANUAL (FUNCTION) (B-64303EN-1)

Edition	Date	Contents	Edition	Date	Contents
02	Aug., 2010	Total revision			
01	May, 2008	———————————————			